DIVISION DE EDUCACION CONTINUA
CURSOS ABIERTOS
XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

MODULO II: ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMOS

del 5 al 10 de junio de 1995.

F E C H A	H O R A R I O	T E M A	P R O F E S O R
Lunes 5	17;00 a 21;00 hrs.	Introducción Análisis Sísmico de sistemas de un grado de libertad	Dr. Octavio A. Rascón Chávez
Martes 6	17;00 a 21;00 hrs.	Análisis sísmico de sistemas discretos de varios grados de libertad	M.I. José L. Trigos Suárez
Miércoles 7	17;00 a 21;00 hrs.	Análisis sísmico. Consideraciones Generales	M.I. José L. Trigos Suárez
Jueves 8 y Viernes 9 Sábado 10	17;00 a 21;00 hrs. 9;00 a 13;00 hrs.	Análisis dinámico Análisis simplificado, método estático, coeficientes sismicos. Método simplificado. Coeficientes sismicos. Torsión.	M.I. Ramón Cervantes

EVALUACION DEL PERSONAL DOCENTE

CURSO: MODULO II: ANALISIS ESTATICO Y DINAMICO DE ESTRUC. SUJETAS A SISM

FECHA: del 5 al 10 de junio de 1995.

CONFERENCISTA	DOMINIO DEL TEMA	USO DE AYUDAS AUDIOVISUALES	COMUNICACION CON EL ASISTENTE	PUNTUALIDAD
Dr. Octavio A. Rascón Chávez				
M.I. José Luis Trigos Suárez				
M.I. Ramón Cervantes				

EVALUACION DE LA ENSEÑANZA

ORGANIZACION Y DESARROLO DEL CURSO	
GRADO DE PROFUNDIDAD LOGRADO EN EL CURSO	
ACTUALIZACION DEL CURSO	
APLICACION PRACTICA DEL CURSO	

EVALUACION DEL CURSO

CONCEPTO	CALIF.
CUMPLIMIENTO DE LOS OBJETIVOS DEL CURSO	
CONTINUIDAD EN LOS TEMAS	
CALIDAD DEL MATERIAL DIDACTICO UTILIZADO	

ESCALA DE EVALUACION: 1 A 10.

1.- ¿LE AGRADO SU ESTANCIA EN LA DIVISION DE EDUCACION CONTINUA?

SI	NO

SI INDICA QUE "NO" DIGA PORQUE.

2.- MEDIO A TRAVES DEL CUAL SE ENTERO DEL CURSO:

PERIODICO EXCELSIOR		FOLLETO ANUAL		GACETA UNAM		OTRO MEDIO	
PERIODICO EL UNIVERSAL		FOLLETO DEL CURSO		REVISTAS TECNICAS			

3.- ¿QUE CAMBIOS SUGERIRIA AL CURSO PARA MEJORARLO?

____

____

4.- ¿RECOMENDARIA EL CURSO A OTRA(S) PERSONA(S)?

SI		NO	

5.- ¿QUE CURSOS LE SERVIRIA QUE PROGRAMARA LA DIVISION DE EDUCACION CONTINUA.

____

____

6.- OTRAS SUGERENCIAS:

____

____

CURSOS ABIERTOS
XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA
MODULO II: ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS
SUJETAS A SISMO
DEL 5 AL 10 DE JUNIO DE 1995
DIRECTORIO DE PROFESORES

M. EN I. RAMON CERVANTES BELTRAN
DIRECTOR GENERAL
CONSTRUCTORA INGENIERIA 2100, S.A. DE C.V.
CALZADA DE LOS CORCELES 64
COL. DEL SUR
01430 MEXICO, D.F.
TEL. 593 25 30

M. EN I. JOSE LUIS TRIGOS SUARES
DIRECTOR GENERAL
TRIGOS INGENIEROS CONSULTORES, S.A.
AV. TASQUEÑA 1818 LOCAL 9
COL. COYOACAN
04280 MEXICO, D.F.
TEL. 689 68 88
FAX 689 66 39

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

CURSOS ABIERTOS.

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

MOD. II " ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMO"

Del 5 al 10 de junio de 1995.

ANALISIS SISMICO DE EDIFICIOS CON FUNDAMENTO EN EL REGLAMENTO DE CONSTRUCCIONES PARA EL DIST.FEDERAL (FCDF87)

ING. RAMON CERVANTES BELTRAN.

Palacio de Minería Calle de Tacuba 5 Primer piso Deleg. Cuauhtémoc 06000 México, D.F. APDO. Postal M-2285
Teléfonos: 512-8955 512-5121 521-7335 521-1987 Fax 510-0573 521-4020 AL 26

PASCUAL LOPEZ GOMEZ
PROFESOR
ENEP ARAGON
AV. RANCHO SECO S/N
BOSQUES DE ARAGON
CD. NEZAHUACOYOTL, EDO. DE MEX.
TEL. 623 08 50

RICARDO MARROQUIN ROSADA
JEFE DE AREA
UNIVERSIDAD IBEROAMERICANA
PROL. PASEO DE LA REFORMA 880
COL. LOMAS DE SANTA FE
DEL. A. OBREGON, MEXICO, D.F.
TEL. 292 25 08

JOSE FCO. ORTEGA LOERA
DIRECTOR
INGS. CONSTRUCTORES, SA.CV.
INSURGENTES NORTE 755-2
COL. SAN SIMON
06920 MEXICO, D.F.
TEL. 597 86 09

CLAUDIA ROJAS SERNA
CERRO DE LAS CRUCES 218
COL. LOS PIRULES
54040 TLANEPANTLA, EDO. DE MEXICO

JAIME RUIZ

JORGE A. LOPEZ RODRIGUEZ
ENCARGADO DEPTO. DE ESTRUC.
COM. DE VIALIDAD Y TRANS.U.
UNIVERSIDAD 800
STA. CRUZ ATOYAC
03310 MEXICO, D.F.
TEL. 688 89 55 EXT. 281

MIGUEL A. MARTINEZ GUERRA
DIRECTOR
ASESORIA Y DISEÑO ESTRUC.
RIO NIAGARA 9-604
COL. CUAUHTEMOC
06500 MEXICO, D.F.
TEL. 208 16 75

MARTIN REYES NORIEGA
S.C.T.

SE LUIS RUBIO MORA
OYECTISTA
TECNICA Y HABITAT
ELIGO ANCONA 86
SANTA MA. LA RIVERA
06400 MEXICO, D.F.
TEL. 541 58 48

ANTONIO SAMPAYO TRUJILLO
UNIDAD HAB. COPILCO
EDIFICIO 1 DEPTO. 106
COL. COPILCO
04350 MEXICO, D.F.
TEL. 658 91 32

1. INTRODUCCION

Uno de los temas del Curso Internacional de Ingeniería Sísmica que cada año organiza la División de Educación Continua de la Facultad de Ingeniería, UNAM, es la cuantificación de las fuerzas que un sismo de diseño le ocasiona a un edificio, de acuerdo con los métodos que recomienda algún código que refleje las experiencias del comportamiento de tales edificacions ante la ocurrencia sistemática de dichos fenómenos naturales de magnitudes significativas, como es el Reglamento de Construcciones para el Distrito Federal vigente (RCDF87).

El hablar de edificios implica una geometría muy especial (trabes, columnas, muros, losas, etc.) construída con determinados materiales (concreto, acero, mamposteria, etc.) que durante su vida útil va a estar sometida a una serie de solicitaciones que tiene que resistir, entre las que se cuenta las debidas a los sismos. Durante el desarrollo de la tecnología que conduce a construír edificaciones seguras y económicas, el ingeniero ha desarrollado una serie de métodos que involucran los conceptos señalados (geometría, material y cargas), que en conjunto conducen al concepto de estructura; y ,desde luego, que el concepto de cargas, a medida que se define con mayor precisión se tiene que relacionar cada vez mas con los otros dos (geometría y material).

El tratar de cuantificar a uno (fuerzas) de los tres conceptos que definen a las estructuras (geometría, material y fuerzas) independientemente de los modelos estructurales del cual forman parte, es prácticamente imposible sin involucrar hipótesis simplificadoras que necesariamente deben conducir a resultados conservadores.

Los métodos basados en hipotesis simplificadoras y modelos estructurales simplificados se utilizaron con mucha frecuencia cuando la herramienta para operarlos consistía únicamente, en calculadora, papel y lápiz. Todavía existen algunos métodos y modelos que aún se utilizan tanto con las herramientas originales como con las computadoras. Es necesario aclarar que la programación de estos métodos es menos integral que los que se desarrollaron para ser utilizados con una computadora.

En este tema se presentan los conceptos que permiten aplicar los métodos que el RCDF87 recomienda para la cuantificación de las fuerzas que un sismo de diseño le ocasiona a un edificio, a fin de determinar los elementos mecánicos y cinemáticos que dicho sismo de diseño provoca y poder así determinar los estados límites de falla y de servicio que el mismo RCDF87 establece para lograr un diseño racional de dichas edificaciones.

2. MODELACION ESTRUCTURAL DE LAS EDIFICACIONES

De acuerdo con el análisis estructural, que es la teoría que involucra a los conceptos de geometría, material y cargas con las leyes de la mecánica newtoniana, se pueden construír modelos que son extraordinariamente simples o bién extraordinariamente refinados, según la herramienta de trabajo (calculadora, computadora, etc) de que se disponga para su manejo. Desde luego que los modelos refinados (grandes geometrías, fuerzas dinámicas, no linealidad geométrica, no linealidad del material, etc.) implican, necesariamente, el uso de la computadora.

Art 189 del RCDF87 establece que: Las fuerzas internas (elementos mecánicos) y las deformaciones (elementos cinemáticos) producidas por las acciones se determinarán mediante un análisis estructural realizado con un método reconocido que tome en cuenta las propiedades de los materiales ante el tipo de cargas que se consideren".

Las normas técnicas complementarias (NTC) para diseño y construcción de estructuras de concreto y de estructuras metálicas del RCDF87, establecen que dichas estructuras se pueden analizar con métodos que supongan un comportamiento elástico, lineal.

Con base en lo anterior el RCDF87 permite utilizar el modelo mas simple del análisis estructural: Material elástico lineal (material de Hooke), desplazamientos pequeños (tensor de deformaciones infinitesimales), que es un modelo matemático lineal basado en la teoría de la elasticidad lineal y la teoría de la mecánica de materiales.

2.1 Representación esquemática

A fin de tener una referencia de los elementos que definen a un edificio, en la Fig 2.1 se representa, de manera esquemática, a los siguientes elementos.

2.1.1 Elementos de la superestructura

De acuerdo con la Fig 2.1 los elementos que conforman a la superestructura son aquéllos que sobresalen del suelo en el que se apoya el edificio, y son:

a) Trabes (elementos barra tridimensionales contenidos en planos horizontales denominadas losas).
b) Columnas (elementos barras tridimensionales contenidos en planos verticales).
c) muros (elementos sólidos tridimensionales contenidos en uno solo o en varios planos verticales).
d) Losas (Elementos tridimensionales contenidos en planos horizontales, idealizados ya como diafragmas flexibles o bien como diafragmas rígidos).

Los elementos de la superestructura se construyen con materiales especificados y controlados por el ingeniero.

2.1.2 Elementos del suelo

El soporte de la estructura lo constituye el suelo, material de dos fase (fase sólida, denominada esqueleto, y fase fluída, generalmente agua y gas) construído de manera natural, por lo que el ingeniero ha desarrollado la tecnología apropiada para su modelación.

2.1.3 Elementos de la cimentación

Los elementos de la cimentación se construyen con materiales especificados y controlados por el ingeniero y pueden ser los siguientes.

a) Contratrabes (elementos barra tridimensionales contenidos en planos horizontales denominadas losas de cimentación, trabes de liga, etc.).
b) Zapatas aisladas o corridas (losas y contratrabes).
c) Muros verticales contenidos en planos verticales.
d) Losas y cascarones (elementos tridimensionales contenidos en una superficie).
e) Pilas y pilotes.

2.2 Elementos estructurales

Con base en los elementos estructurales de las edificaciones indicados de manera esquemática en la sección 2.1, en esta sección o un cuadrilátero (con cuatro o mas puntos nodales), según se indica en la Fig 2.2. Además de los tres componentes de desplazamiento correspondientes a los elementos losas se le adicionan los tres desplazamientos del elemento membrana(dos lineales contenidos en su superficie y uno angular normal a su superficie). Las ecuaciones de equilibrio se establecen mediante alguna de las teorías de la mecánica del medio continuo (como la teoría de la elasticidad lineal) y para su solución se utiliza el MEF.

2.2.5 Diafragmas flexibles

Los diafragmas son elementos planos (en los edificios) que unen a varios elementos estructurales que los obliga a desplazarse en conjuto, como si fuera una membrana. Desde luego que existen desplazamientos relativos entre los elementos unidos por el diafragma. A cada punto nodal de los elementos estructurales contenido en el diafragma le coresponden dos desplazamientos lineales y un angular, que desde luego son independientes para cada punto nodal (Fig 2.2). Los diafragmas flexibles se modelan mediante el elemento finito cascarón del inciso 2.2.4.

2.2.6 Diafragmas rígidos

Cuando los desplazamientos relativos entre los elementos unidos por el diafragma (descrito en el inciso 2.2.5) son pequeños y se pueden considerar nulos, se dice que el diafragma es rígido y, por tanto, los desplazamientos de los puntos nodales contenidos en el diafragma son linealmente dependientes de los tres desplazamientos del diafragma (dos lineales y un angular). Desde luego que el número de desplazamientos independientes del diafragma rígido (únicamente tres, Fig 2.2)) resulta ser mucho menor que el correspondiente a los del diafragma flexible (seiss por el número de puntos nodales contenidos en dicho diafragma).

2.3 Modelos estructurales

Con el ensamble de los elementos estructurales descritos en el inciso 2.2 se puede construír una gran variedad de modelos estructurales que se pueden utilizar en el análisis estructural de los edificios. Independientemente de los elementos estructurales que participan en su ensamble, las ecuaciones de equilibrio de los modelos estructurales sometidos a cargas estáticas resultan ser.

$$\vec{K}\vec{U} = \vec{F} \tag{2.3}$$

Los vectores y la matriz de los modelos estructurales dados por la Ec 2.3 se denominan.

El número de componentes de los vectores de la estructura (Ec 2.4)

$\vec{U}$ = Vector de desplazamientos de la estructura (desconocido)

$\vec{F}$ = Vector de fuerzas de la estructura (conocido) (2.4)

$\vec{K}$ = Matriz de rigideces de la estructura (conocida)

es igual al número de componentes de desplazamiento (lineales y angulares) desconocidos, linealmente independientes, de los puntos nodales de la estructura (grados de libertad de la estructura). Los modelos estructurales mas comunes se describen a continuación.

2.3.1 Marcos tridimensionales

Es un modelo estructural formado esclusivamente con los elementos barras barra descritos en el inciso 2.2.1. Necesariamente debe contener barras tridimensionales, pero también pueden existir combinaciones de barras planas, barras de retícula de entrepiso y barras axiales.

2.3.2 Muros tridimensionales

Este modelo se construye con el ensamble de elementos sólidos bidimensionales (inciso 2.2.2), elementos placas planas (inciso 2.2.3) y elementos cascarones (inciso 2.2.4), según el tipo de carga que actúa en sus respectivas regiones.

2.3.3 Muromarcos tridimensionales

El modelo de muromarcos tridimensionales es una combinación de los modelos marcos tridimensionales y muros tridimensionales.

2.3.4 Marcos planos

Este modelo es un caso particular de los marcos tridimensionales y se obtiene mediante el ensamble de barras planas, por lo que su geometria y cargas están contenidas en un plano.

2.3.5 Muros planos

Este modelo es un caso particular de los muros tridimensionales y se obtiene mediante el ensamble de elementos sólidos bidimensionales, por lo que su geometria y cargas están contenidas en un plano.

2.3.6 Muromarcos planos

El modelo de muromarcos planos es una combinación de los modelos marcos planos y muros planos.

se resumen los conceptos formales de tales elementos estructurales en relación con su participación en la construción de las ecuaciones de equilibrio de la edificación.

El método mas versatil y poderoso para formular, resolver y manejar las ecuaciones de equilibrio de las estructuras, es el método de las rigideces o de los desplazamientos (para los elementos barras, asociados a las estructuras esqueletales o marcos) y el método del elemento finito en su formulación de los desplazamientos (para los elementos sólidos bidimensionales, placas planas y cascarones de las estructuras denominadas continuas). La versatilidad y poderío de los métodos anteriores están asociados a su adecuación al uso de las computadoras.

Las ecuaciones de equilibrio de los elementos estructurales se establecen en términos de los puntos nodales que se requieren para definir su geometría. A los puntos nodales de cada elemento finito le corresponden diferentes grados de libertad (número de componentes de desplazamiento lineales y agulares).

Para el caso de fuerzas estáticas, las ecuaciones de quilibrio de cada elemento estructural se puede escribir, de manera general, de la siguiente manera:

$$\begin{aligned} \vec{f}^{e} &= \vec{f}^{0} + \bar{k}\vec{u} \\ &= \vec{f}^{0} + \vec{f}^{u} \end{aligned} \qquad (2.1)$$

donde los vectores y la matriz de la ecuación anterior están asociados a los elementos mecánicos y cinemáticos de los puntos nodales del elemento estructural, y los nombres mas comunes que reciben son los siguientes.

$$\begin{aligned} \vec{f}^{e} &= \text{Vector de fuerzas equilibrantes} \\ \vec{f}^{0} &= \text{Vector de fuerzas de empotramiento} \\ \vec{f}^{u} = \bar{k}\vec{u} &= \text{Vector de fuerzas de desplazamiento} \\ \bar{k} &= \text{Matriz de rigideces} \\ \vec{u} &= \textit{Vector de desplazamientos} \end{aligned} \qquad (2.2)$$

En las Ec 2.1 y 2.2 la magnitud y el número de los componentes de los vectores y de la matriz dependen del número de puntos nodales y de sus correspondientes grados de libertad que definen al elemento estructural.

2.2.1 Elementos barra

Son elementos tridimensionales para representar a trabes, columnas, contratrabes, pilas y pilotes (Fig 2.2). Geométricamente bastan dos puntos nodales que definen un eje (casi siempre recto) y sus secciones transversales (casi siempre constantes y, por tanto, con una basta). Sus ecuaciones de equilibrio se obtienen con base en la teoría de la mecánica de materiales y para su integración no se requiere del método del elemento finito (MEF), para las barras de eje recto y sección constante. A cada punto nodal se le consideran seis grados de libertad, tres lineales y tres angulares. Se presentan caso particulares como son las barras planas con tres grados de libertad por nudo (dos lineales y un angular), las barras de retícula de entrepiso con tres grados de libertad por nudo (uno lineal y dos angulares), las barras de armaduras (barras axiales o barras doblemente articuladas) con tres (tridimensionales) y dos (bidimensionales) grados de libertal por nudo (que son desplazamientos lineales, ya que los angulares son linealmente dependientes por corresponder a articulaciones). En general, los vectores tienen seis componentes.

2.2.2 Elementos sólidos bidimensionales (muros planos)

Son elementos tridimensionales que únicamente pueden soportar cargas y desplazamientos contenidos en su superficie media (plana). Geométricamente se pueden definir mediante un triángulo (tres o mas puntos nodales) o un cuadrilátero (con cuatro o mas puntos nodales), según se indica en la Fig 2.2. A cada punto nodal normalmente se le asignan dos componentes de desplazamiento lineal. Las ecuaciones de equilibrio se establecen mediante alguna de las teorías de la mecánica del medio continuo (como la teoría de la elasticidad lineal) y para su solución se utiliza el MEF.

2.2.3 Elementos placas planas (losas)

Son elementos tridimensionales que generalmente se utiizan para soportar cargas transversales a su superficie media (plana). Geométricamente se pueden definir mediante un triángulo (tres o mas puntos nodales) o un cuadrilátero (con cuatro o mas puntos nodales), según se indica en la Fig 2.2. A cada punto nodal normalmente se le asignan tres componentes de desplazamiento (uno lineal y angulares). Las ecuaciones de equilibrio se establecen mediante alguna de las teorías de la mecánica del medio continuo (como la teoría de la elasticidad lineal) y para su solución se utiliza el MEF.

2.2.4 Elementos cascarones (muros tridimensionales)

Son elementos tridimensionales que generalmente se utiizan para soportar tanto cargas transversales a su superficie media (losa) como cargas contenidas en su superficie (membrana). Geométricamente se pueden definir mediante un triángulo (tres o mas puntos nodales) marcos y muromarcos tridimensionales se condensan las ecuaciones de los grados de libertad que no están asociados a los tres desplazamientos del diafragma rígido, mediante un triangulación parcial. El número de ecuaciones de equilibrio de este modelo es igual a tres veces el número de diafragmas rígidos, que es mucho menor que el modelo descrito en el inciso 2.4.1 y también menor que el del inciso 2.4.2 en caso de existir muros en el edificio.

Al considerar varias subestructuras unidas con el diafragma rígido, existen elementos que forman parte de dos o mas subestructuras que, desde luego, se proporcionan desplazamientos independientes, a menos que se establezca un criterio que reduzca este problema característico de este modelo. Otra forma de evitar este problema es considear una sola subestructura que resulta del tamaño del edificio.

2.4.4 Subestructuras formadas con marcos y muromarcos planos unidos con diafragmas rígidos (TABS)

Este modelo corresponde a la versión original del modelo anterior (inciso 2.4.3) en donde se utilizan como subestructuras a las estructuras planas (marcos, muros y muromarcos), somo se muestra en la Fig 2.6. La sigla TABS se refiere a: Three dimensional Analysis of Building System.

En este modelo siembre existe la incompatibilidad de los desplazamientos en los elementos comunes de las estructuras planas, a menos que se establezca un criterio que reduzca este problema.

2.4.5 Subestructuras formadas con rigideces de entrepiso (resortes) unidas con diafragmas rígidos

Este modelo es una simplificación del modelo anterior (inciso 2.4.4) en donde las subestructuras resultan ser las rigideces de entrepiso asociadas a cada muro o muromarco, según se indica en la Fig 2.7.

Las rigideces de entrepiso se consideran que están orientadas en dos direcciones ortogonales que forman dos modelos estructurales (unidireccionales) independientes, según se muestra en la Fig 2.9. Los grados de libertad de cada modelo estructural independiente estan formados por los desplazamientos horizontales de cada diafragma en la dirección que le corresponde al modelo (el número de ecuaciones es igual al número de diafragmas rígidos).

Una vez calculadas las fuerzas sismicas asociadas a cada modelo unidireccional independiente, se procede a unir cada diafragma rígido aislado con las rigideces de entrepiso que les subyace y se le aplica la fuerza cortante de dicho entrepiso. La fuerza cortante es la que se distribuye entre las rigideces de entrepiso que subyacen al diafragma, al considerar el equilibrio de cada diafragma independiente de los demás.

Con la fuerza cortante que a cada rigidez de entrepiso le corresponde, se cuantifican las fuerzas sismicas de cada nivel, que son las que se aplican a las estructuras planas correspondientes a las rigideces de entrepiso (marcos, muros o muromarcos).

2.4.6 Método simplificado del RCDF87

En este método, las Normas Técnicas Complementarias (NTC) para diseño y construcción de estructuras de mampostería establece que, es admisible considerar que la fuerza cortante que toma cada muro es proporcional a su área transversal e ignorar los efectos de torsión. Las fuerzas sísmicas con las que se obtienen las fuerzas cortantes se cuantifican de manera independiente del modelo estructural del edificio.

2.3.7 Rigideces de entrepiso (resortes)

Este modelo estructural únicamente sirve para simplificar el análisis de marcos planos ante fuerzas horizontales. Con algunas hipótesis simpificadoras se hace extensivo a muros planos y a muromarcos planos.

Como se muestra en la Fig 2.3, la estructura plana original (marco, muro o muromarco) se reemplaza por una estructura a base de resortes. La constante del resorte, denominada rigidez de entrepiso, se cuantifica de acuerdo con la siguiente expresión.

$$k_i = \frac{V_i}{\Delta u_i} = \text{Rigidez de entrepiso} \tag{2.5}$$

Los elementos de la Ec 2.5 se muestran en la Fig 2.8 y se definen como.

$$\begin{aligned} \Delta u_i &= \text{Desplazamiento relativo del i-ésimo entrepiso} \\ &= u_i - u_{i-1} \\ u_i &= \text{Desplazamiento horizontal del i-ésimo nivel} \\ u_{i-1} &= \text{Desplazamiento horizontal del (i-1)-ésimo nivel} \\ V_i &= \text{Fuerza cortante del i-ésimo entrepiso} \end{aligned} \tag{2.6}$$

Desde luego que en la Ec 2.5 no se conocen los desplazamientos horizontales de los niveles y para cuantificar los valores de las rigideces de entrepiso se hacen hipótesis respecto a los desplazamientos angulares y fuerzas cortantes en los entrepisos y niveles adyacentes (como es el caso de las fórmulas de Wilbur).

Por supuesto que las rigideces de entrepiso se pueden cuantificar mediante el uso de la computadora al estimar las fuerzas horizontales que actúan en las estructuras planas, pero resulta mucho menos eficiente que utilizar los métodos de análisis que existen y que fueron diseñados para ser manejados por una computadora.

2.4 Modelos estructurales para el análisis de edificios ante fuerzas sísmicas

Un concepto básico para cuantificar las fuerzas sismicas en las edificaciones es el modelo estructural utilizado. En este inciso se describen, de manera esquemática, los modelos estructurales que se utilizan en el análisis sísmico de las edificaciones.

2.4.1 Marcos y muromarcos tridimensionales unidos con diafragmas flexibles

El modelo estructural del edificio se forma con los modelos estructurales correspondientes a marcos y muromarcos tridimensionales (incisos 2.3.1 y 2.3.3) unidos mediante un diafragma flexible (inciso 2.6), según se muestra en la Fig 2.4.

El número de ecuaciones de equilibrio está asociado a los componentes de desplazamiento (lineales y angulares) linealmente independientes de los puntos nodales del edificio, que aún para edificios relativamente pequeños resulta ser un número grande comparado comparado con otros modelos. Este modelo puede provocar problemas de aproximación debido a que la modelación de la rigidez en el plano del diafragma resulta ser muy grande.

Desde luego que este modelo estructural únicamente se puede manejar con una computadora y se construye al utilizar los programas de propósitos generales basados en el MEF (NISA, SAP90, etc.).

2.4.2 Marcos y muromarcos tridimensionales unidos con diafragmas rígidos

Algunos programas de propósitos generales basados en el MEF (SAP90) contemplan la posibilidad de hacer que puntos nodales contenidos en un diafragma sean linealmente dependientes respecto a un punto (centro de masas). Esto obliga a que cada diafragma tenga tres grados de libertad, lo que reduce significativamente el número de ecuaciones que genera el modelo del inciso anterior (inciso 2.4.1) y elimina los problemas de aproximación debido a las rigideces grandes en el plano del diafragma.

2.4.3 Subestructuras formadas con marcos y muromarcos tridimensionales unidos con diafragmas rígidos(ETABS)

Existen programas de computadora de propósitos especiales (La sigla ETABS se refiere a: Extended Three dimensional Analysis of Building System) en los que se toma en cuenta las particularidades de los elementos que conforman a un edificio (muros, trabes, columnas, juntas, diafragma rígido).

La construcción de este modelo se basa en considerar a los marcos y muromarcos tridimensionales como una subestructura, según se observa en la Fig 2.5. De las ecuaciones de equilibrio de los

3. PARAMETROS QUE DEFINEN LA MAGNITUD DE LAS FUERZAS SISMICAS

A continuación se resumen los parámetros que el Reglamento de Construcciones para el Distrito Federal (RCDF87) considera para cuantificar la magnitud de las fuerzas que un sismo de diseño ocasiona a una estructura.

3.1 Uso de las edificaciones

De acuerdo con el RCDF87 se tiene que:

Art 174. Para los efectos de este Título (VI, Seguridad estructural de las construcciones) las construcciones se clasifican en los siguientes grupos:

I. GRUPO A. Construcciones cuya falla estructural podría causar:

La pérdida de un número elevado de vidas, o

Pérdidas económicas o culturales excepcionalmente altas, o

Que constituyen un peligro significativo por contener sustancias tóxicas o explosivas,

Así como construcciones cuyo funcionamiento es esencial a raíz de una emergencia urbana como:

Hospitales y escuelas,
Estadios,
Templos,
Salas de espectáculos y hoteles que tengan salas de

reunión que pueden alojar mas de 200 personas;
Gasolinerías,
Depósitos de sustancias inflamables o tóxicas,
Terminales de transporte,
Estaciones de bomberos,
Subestaciones eléctricas y centrales telefónicas y de telecomunicaciones,
Archivos y registros públicos de especial importancia a juicio del DDF,
Museos,
Monumentos y
Locales que alojen equipo especialmente costoso

II. GRUPO B. Construcciones comunes destinadas a:

Vivienda,
Oficinas y locales comerciales,
Hoteles y
Construcciones comerciales e industriales no incluídas en el grupo A, las que se subdividen en:

a) SUBGRUPO B1. Construcciones de más de 30 m de altura o con más de 6,000 m² de área total construida, ubicadas en las zonas I y II según se define en el artículo 175, y
Construcciones de más de 15 m de altura o 3,000 m² de área total construída, en zona III, y

b) SUBGRUPO B2. Las demás de este grupo.

3.2 Coeficiente sísmico

De acuerdo con el RCDF87 se tiene:

Art 206. El coeficiente sísmico, c, es el cociente de la fuerza cortante horizontal que debe considerarse que actúa en la base de la construcción por efecto del sismo (Vo) entre el peso de ésta sobre dicho nivel (Wo).

Con este fin se tomará como base de la estructura el nivel a partir del cual sus desplazamientos con respecto al terreno circundante comienzan a ser significativos. Para calcular el peso total se tendrán en cuenta las cargas muertas y vivas que correspondan según los capítulos IV Y V de este Título (VI).

El coeficiente sísmico para las construcciones clasificadas como grupo B en el artículo 174 se tomarán los siguientes valores:

Zona No.	Coeficiente sísmico (c)
I	0.16
II	0.32
III	0.40

A menos que se emplee el método simplificado de análisis en cuyo caso se aplicarán los coeficientes que fijen las NTC, y a excepción de las zonas especiales en las que dichas NTC especifiquen otros valores de c.

Para las estructuras del Grupo A se incrementará el coeficiente sísmico en 50 por ciento.

De acuerdo con lo anterior se puede escribir

$$c = \frac{V_0}{W_0} = \text{Coeficiente sísmico}$$

donde:

$$V_0 = \sum_{i=1}^{No\ niv} F_i = \text{Fuerza cortante en la base}$$

$$W_0 = \sum_{i=1}^{No\ niv} W_i = \text{Peso de la construcción} \qquad (3.1)$$

F_i = Fuerza sísmica en el i-ésimo nivel

W_i = Peso de la construcción en el i-ésimo nivel

3.3 Zonificación sísmica

De acuerdo con el RCDF87 se tiene

Art 175. Para fines de estas disposiciones, el DF se considera dividido en las zonas I, II y III, dependiendo del tipo de suelo.

Las características de cada zona y los procedimientos para definir la zona que corresponde a cada predio se fijan en el capítulo VII (Diseño de cimentaciones) de este Título (VI. Seguridad estructural de las construcciones).

Art 219. Para fines de este Título (VI) el DF se divide en tres zonas con las siguientes características generales:

Zona I. LOMAS, formadas por rocas o suelos generalmente firmes que fueron depositados fuera del ambiente lacustre, pero en los que pueden existir, superficialmente o incrustados, depósitos arenosos en estado suelto o cohesivos relativamente blandos. En esta zona, es frecuente la presencia de oquedades en rocas y de cavernas y túneles excavados en suelos para explotar minas de arena.

Zona II. TRANSICION, en la que los depósitos profundos se encuentran a 20 m de profundidad o menos, y que está constituída predominantemente por estratos arenosos y limoarenosos intercalados con capas de arcilla lacustre; el espesor de éstas es variable entre decenas de centímetros y pocos metros, y

Zona III. LACUSTRE, integrada por potentes depósitos de arcilla altamente compresible, separados por capas arenosas con contenido diverso de limo o arcilla. Estas capas arenosas son de consistencia firme a muy dura y de espesores variables de centímetros a varios metros.

Los depósitos lacustras suelen estar cubiertos superficialmente por suelos aluviales y rellenos artificiales; el espesor de este conjunto puede ser superior a 50 m.

La zona a que corresponda un predio se determinará a partir de las investigaciones que se realicen en el subsuelo del predio objeto de estudio, tal y como lo establecen las NTC. En caso de construcciones ligeras o medianas, cuyas características se definirán en dichas normas (NTC para cimentaciones) podrá determinarse la zona mediante el mapa incluído en las mismas (ver fig 1 NTC para cimentaciones), si el predio está dentro de la porción zonificada; los predios ubicados a menos de 200 m de las fronteras entre dos de las zonas antes descritas se supondrán ubicados en la más desfavorable.

Art 220. La investigación del subsuelo del sitio mediante exploración de campo y pruebas de laboratorio debe ser suficiente para definir de manera confiable:

Los parámetros de diseño de la cimentación.
La variación de los mismos en la planta del predio.
Los procedimientos de construcción.
Además deberá ser tal que permita definir:

I. En la zona I a que se refiere el artículo 219 del RCDF, si existen en ubicaciones de interés materiales sueltos superficiales, grietas, oquedades naturales o galerías de minas, y en caso afirmativo su apropiado tratamiento, y.

II. En las zonas II y III del artículo mencionado en la fracción anterior, la existencia de restos arqueológicos, cimentaciones antiguas, grietas, variaciones fuertes de estratigrafía, historia de carga del predio o cualquier otro factor que pueda originar asentamientos diferenciales de importancia, de modo que todo ello pueda tomarse en cuenta en el diseño.

Las NTC para cimentaciones en su capítulo 2 (Investigaciones del subsuelo) establecen en la tabla I los requisitos mínimos para la investigación del subsuelo para las construcciones ligeras o medianas de poca extensión y con excavaciones someras, y para las construcciones pesadas, extensas o con excavaciones profundas.

Las NTC para sismo en su capítulo 3 (Espectros para diseño sísmico) establecen que el coeficiente, c, que se obtiene del Art 206 del RCDF87 salvo en la parte sombreada de la zona II (ver fig. 3.1 de dichas NTC) toma los ssiguientes valores:

$c = 0.4$ para las estructuras del grupo B, y

$c = 0.6$ para las estructuras del grupo A.

3.4 Condiciones de regularidad

De acuerdo con las NTC para el diseño por sismo, en su capítulo 6, para que una estructura pueda considerarse regular debe satisfacer los siguientes requisitos:

1. Su planta es sensiblemente simétrica con respecto a dos ejes ortogonales por lo que toca a masas, así como a muros y otros elementos resistentes.

2. La relación de su altura a la dimensión menor de su base no pasa de 2.5.

3. La relación de largo a ancho de la base no excede de 2.5.

4. En la planta no tiene entrantes ni salientes cuya dimensión exceda de 20 por ciento de la dimensión de la planta medida paralelamente a la dirección que se considera de la antrante o la saliente.

5. En cada nivel tiene un sistema de techo o piso rígido y resistente .

6. No tiene aberturas en sus sistemas de techo o piso cuya dimensión exceda de 20 por ciento de la dimensión en planta medida paralelamente a la dimensión que se considere de la abertura, las áreas huecas no ocasionan asimetrías significativas ni difieren de posición de un piso a otro y el área total de aberturas no excede en ningún nivel de 20 por ciento del área de la planta.

7. El peso de cada nivel, que incluye la carga viva que debe considerarse para diseño sísmico, no es mayor que el del piso inmediato inferior ni, excepción hecha del último nivel de la construcción, es menor que 70 por ciento de dicho peso.

8. Ningún piso tiene un área, delimitada por los paños exteriores de sus elementos resistentes verticales, mayor que la del piso inmediato inferior ni menor que 70 por ciento de ésta. Se exime de este último requisito únicamente al último piso de la construcción.

9. Todas las columnas están restringidas en todos los pisos en dos direcciones ortogonales por diafracmas ortogonales y por trabes o losas planas.

10. La rigidez al corte de ningún entrepiso excede en más de 100 por ciento a la del entrepiso inmediatamente inferior.

11. En ningún entrepiso la excentricidad torsional calculada estáticamente, e_s, excede del 10 por ciento de la dimensión en planta de ese entrepiso medida paralelamente a la excentricidad mencionada.

NOTA: En el capítulo 4 (Reducción de fuerzas sísmicas) de las NTC para diseño por sismo (inciso 4.4.2 de estas notas) se especifica que: "... En el diseño sísmico de las estructuras que no satisfacen las condiciones de regularidad que fija la sección 6 de estas normas, se multiplicará por 0.8 el valor de Q'."

3.5 Factor de comportamiento sísmico

De acuerdo con el RCDF87 se tiene que

Art 207. Cuando se aplique el método estático o un método dinámico para análisis sísmico, podrán reducirse con fines de diseño las fuerzas sísmicas calculadas, empleando para ello los criterios que las NTC, en función de las características estructurales y del terreno. Los desplazamientos calculados de acuerdo con estos métodos, empleando las fuerzas sísmicas reducidas, deben multiplicarse por el factor de comportamiento sísmico que marquen dichas Normas.

Los coeficientes que especifique las NTC para la aplicación del método simplificado de análisis tomarán en cuenta todas las reducciones que procedan por los conceptos mencionados. Por ello las fuerzas sísmicas calculadas por este método no deben sufrir reducciones adicionales.

De acuerdo con las NTC para sismo del RCDF87 en su capítulo 5, los valores de los factores del comportamiento sísmico, Q, se especifican a continuación:

I. Se usará Q=4 cuando se cumplan los requisitos siguientes:

1. La resistencia en todos los entrepisos es suministrada exclusivamente
   - Por marcos no contraventeados de acero o concreto reforzado, o bien
   - Por marcos contraventeados o con muros de concreto reforzado en los que en cada entrepiso los marcos son capaces de resistir, sin contar muros ni contravientos, cuando menos 50 por ciento de la fuerza sísmica actuante.

2. Si hay muros ligados a la estructura en la forma espoecificada en el caso I del artículo 204 del RCDF87, éstos se deben tener en cuenta en el análisis, pero su contribución a la capacidad ante fuerzas laterales sólo se tomará en cuenta si estos muros son de piezas macizas, y los marcos, sean o no contraventeados, y los muros de concreto reforzado son capaces de resistir al menos 80 por ciento de las fuerzas laterales totales sin la contribución de los muros de mampostería.

3. El mínimo cociente de la capacidad resistente de un entrepiso entre la acción de diseño no difiere en más de 35 por ciento del promedio de dichos cocientes para todos los entrepisos. Para verificar el cumplimiento de este requisito, se calculará la capacidad resistente de cada entrepiso teniendo en cuenta todos los elementos que puedan contribuír a la resistencia, en particular los muros que se hallen en el caso I a que se refiere el artículo 204 del Reglamento.

4. Los marcos y muros de concreto reforzado cumplen con los requisitos que fijan las normas técnicas complementarias correspondientes para marcos y muros dúctiles.

5. Los marcos rígidos de acero satisfacen los requisitos para marcos dúctiles que fijan las normas técnicas complementarias correspondientes.

II. Se adoptará Q=3 cuando se satisfacen las condiciones 2,4 y 5 del caso I y en cualquier entrepiso dejan de satisfacerse las condiciones 1 ó 3 especificadas para el caso I pero la resistencia en todos los entrepisos es suministrada:

Por columnas de acero o de concreto reforzado con losas planas,
Por marcos rígidos de acero,
Por marcos de concreto reforzado,
Por muros de concreto reforzado,
Por combinaciones de muros de concreto reforzado y por marcos o por diafragmas de madera contrachapada.

Las estructuras con losas planas deberán cumplir los requisitos que sobre el particular marcan las normas técnicas complementarias para estructuras de concreto.

III. Se usará Q=2 cuando la resistencia a fuerzas laterales es suministrada

Por losas planas con columnas de acero o de concreto reforzado,
Por marcos de acero o de concreto reforzado, contraventeados o no,
Por muros o columnas de concreto reforzado,

que no cumplen en algún entrepiso lo especificado por los casos I y II de esta sección, o

Por muros de mampostería de piezas macizas confinados por castillos, dalas, columnas o trabes de concreto reforzado o de acero que satisfacen los requisitos de las normas técnicas complementarias respectivas, o diafragmas construidos con duelas inclinadas o por sistemas de muros formados por duelas de madera horizontales o verticales combinados con elementos diagonales de madera maciza.

También se usará Q=2 cuando la resistencia es suministrada por elementos de concreto prefabricado o presforzado, con la excepciones que sobre el particular marcan las normas técnicas complementarias para estructuras de concreto.

IV. Se usará Q=1.5 cuando la resistencia a fuerzas laterales es suministrada en todos los entrepisos

Por muros de mampostería de piezas huecas, confinados o con refuerzo interior, que satisfacen los requisitos de las normas técnicas complementarias respectivas, o
Por combinaciones de dichos muros con elementos como los descritos para los casos II y III, o por marcos y armaduras de madera.

V. Se usará Q=1 en estructuras cuya resistencia a fuerzas laterales es suministrada al menos parcialmente por elementos o materiales de los arriba especificados, a menos que se haga un estudio que demuestre, a satisfacción del Departamento del Distrito Federal, que se puede emplear un valor más alto que el que aquí se especifica.

En todos los casos se usará para toda la estructura en la dirección de análisis el valor mínimo de Q que corresponde a los diversos entrepisos de la estructura en dicha dirección.

El factor Q puede diferir en las dos direcciones ortogonales en que se analiza la estructura, según sean las propiedades de ésta en dichas direcciones.

3.5.1 Condiciones para marcos dúctiles de concreto

Con base en los puntos I.4 y II del inciso 3.5, se reproduce el Capítulo 5, Marcos dúctiles, de las NTC para diseño y construción de estructuras de concreto del RCDF87.

3.5.1.1 Requisitos generales

Los requisitos de este capítulo se aplican a marcos colados en el lugar, diseñados por sismo con un factor de comportamiento sísmico, Q=4. También se aplican a los marcos de estructuras coladas en el lugar diseñadas con Q=4, formadas por marcos y muros de concreto reforzado que cumplan con el inciso 4.5.2 (de las NTC para diseño y construcion de estructuras de concreto del RCDF87), que debe incluír el inciso b) de esa sección, o marcos y contravientos que cumplan con el inciso 4.6 (de las NTC para diseño y construcion de estructuras de concreto del RCDF87), en las que la fuerza cortante resistida por los marcos sea, al menos, el 50 porciento de la total y, asímismo, a los marcos de estructuras coladas en el lugar, diseñadas con Q=3 y formadas por marcos y muros o contravientos que cumplan con el inciso 4.5.2 (de las NTC para diseño y construcion de estructuras de concreto del RCDF87), que debe incluír el inciso b) de esa sección, o marcos y contravientos que cumplan con el inciso 4.5.2 (de las NTC para diseño y construcion de estructuras de concreto del RCDF87), que debe incluír el inciso b) de esa sección, o el inciso 4.6 (de las NTC para diseño y construcion de estructuras de concreto del RCDF87), en las que la fuerza cortante resistida por los marcos sea menor que el 50 porciento de la total. En todos los casos anteriores, los requisitos se aplican también a los elementos estructurales de la cimentación.

Sea que la estructura esté formada sólo de marcos o de marcos y muros o contravientos, ningún marco se debe diseñar para resistir una fuerza cortante horizontal menor que el 25 porciento de la que le correspondería si trabajara aislado del resto de la estructura.

La resistencia especificada del concreto, f'_c, no debe ser menor de 200 kg/cm^2.

Las barras de refuerzo deben ser corrugadas de grado no mayor que el 42 y deben cumplir con los requisitos de las normas NOM-B6. Además, las barras longitudinales de vigas y columns deben tener fluencia definida, bajo un esfuerzo que no exceda al esfuerzo de fluencia especificadoen mas de 1300 kg/cm^2, y su resistencia real debe ser, al menos, igual a 1.25 veces su esfuerzo real de fluencia.

Se deben aplicar las disposiciones de estas normas (NTC para diseño y construcion de estructuras de concreto del RCDF87) que no se modifiquen en este capítulo.

3.5.1.2 Miembros a flexión

Los requisitos de este inciso se aplican a miembros principales que trabajan esencialmente a flexión. Se incluyen vigas y aquellas columns con cargas axiales pequeñas, tales que.

$$P_u \leq 0.1 A_g f'_c \tag{3.2}$$

3.5.1.2.1 Requisitos geométricos

a) El claro libre no debe ser menor que cuatro veces el peralte efectivo.

b) En sistemas de viga y losa monolítica, la relación entre la separación de apoyos que eviten el pandeo lateral y el ancho de la viga no debe exceder de 30.

c) La relación entre el peralte y ancho no debe ser mayor que 3.

d) El ancho de la viga no debe ser menor de 25 cm, ni debe exceder al ancho de las columnas a las que llega.

e) El eje de la viga no debe separsrse horizontalmente del eje de la columna mad de un décimo de la dimensión transversal de la columna normal a la viga.

3.5.1.2.2 Refuerzo longitudinal

En toda sección se debe disponer de refuerzo tanto en el lecho inferior como en el superior. En cada lecho el área de refuerzo no debe ser menor que.

$$0.7\sqrt{f'_c}\frac{bd}{f_y} \tag{3.3}$$

y debe constar, al menos, por dos barras corridas de 12.7 mm de diámetro (No 4). El área de acero a tensión no debe exceder del 75 por ciento de la correspondiente a la falla balanceada de la sección.

El momento resistente positivo en la unión con un nudo no debe ser menor que la mitad del momento resistente negativo que se suministre en esa sección. En ninguna sección a lo largo del miembro el momento resistente negativo, ni el resistente positivo, deben ser menores que la cuarta parte del máximo momento resistente que se tenga en los extremos.

En las barras para flexión se permiten traslapes solo si en la longitud del traslape se suministra refuerzo transversal de confinamiento (refuerzo helicoidal o estribos cerrados); el paso o la separación de este refuerzo no debe ser mayor que 0.25 d, ni que 10 cm. Las uniones por traslapes no se permiten en los casos siguientes:

a) Dentro de los nudos
b) En una distancia de dos veces el peralte del miembro, medida desde el paño del nudo, y
c) En aquellas zonas donde el análisis indique que se forman articulaciones plásticas.

Con el refuerzo longitudinal pueden formarse paquetes de dos barras cada uno.

Se permiten uniones soldadas o con dispositivos mecánicos, que cumplan con los requisitos del inciso 3.9 (NTC para diseño y construcion de estructuras de concreto del RCDF87), a condición de que en toda sección de unión, cuando mucho, se unan barras alternadas y que las uniones de barras adyacentes no disten entre sí menos de 60 cm en la dirección longitudinal del miembro.

3.5.1.2.3 Refuerzo transversal para confinamiento

Se deben sumministrar estribos cerrados de, al menos, 7.9 mm de diámatro (No 2.5) que cumplan con los requisitos de los párrafos que siguen, en las zonas siguientes:

a) En cada extremo del miembro sobre una distancia de dos peraltes medida a partir del paño del nudo, y
b) En la porción del elemento que se halle a una distancia igual a dos peraltes (2h) de toda sección donde se suponga, o el análisis lo indique, que se va a formar una articulación plástica (si la articulación se forma en una sección intermedia, los dos peraltes se deben tomar a cada lado de la sección).

El primer estribo se debe colocar a no mas de 5 cm de la cara del miembro de apoyo. La separación de los estribos no debe exceder los valores siguientes:

a) 0.25 d
b) Ocho veces el diámetro de la barra longitudinal mas delgada
c) 24 veces el diámetro de la barra del estribo
d) 30 cm

Los estribos a que se refiere esta sección deben ser cerrados, de una pieza, y deben rematar en una esquina con dobleces de 135 grados, seguidos de tramos rectos de no menos de 10 diámetros de largo. En cada esquina del estribo debe quedar, al menos, una barra longitudinal. Los radios de doblez deben cumplir con los requisitos del inciso 3.8 (NTC para diseño y construcion de estructuras de concreto del RCDF87). La localización del reamte del estribo debe alternarse uno a otro.

En las zonas definidas en el primer párrafo de esta sección, las barras longitudinales de la periferia deben tener soporte lateral que cumpla con el inciso 3.3 (NTC para diseño y construcion de estructuras de concreto del RCDF87).

Fuera de las zonas definidas en el primer párrafo de esta sección, la separación de los estribos no debe ser mayor que 0.5d a todo lo largo. En toda la viga la separación de estribos no debe ser mayor que la requerida por fuerza cortante.

3.5.1.2.4 Requisitos para fuerza cortante

Los elementos que trabajan principalmente a flexión se deben dimensionar de manera que no se presente falla por cortante antes que puedan formarse las articulaciones plásticas en sus extremos. Para ello, la fuerza cortante de diseño se obtiene del equilibrio del miembro entre caras de apoyo; se debe suponer que en los extremos actúan momentos del mismo sentido valuados con las propiedades del elemento en esas secciones, sin factores de reducción, y con el esfuerzo en el acero de tensión, al menos, igual a 1.25 f_y. A lo largo del miembro deben actuar las cargas correspondientes multiplicadas por el factor de carga.

Como opción, pueden dimensionarse con base en la fuerza cortante de diseño obtenida del análisis, si el factor de resistencia F_R, se le asigna un valor de 0.6, en lugar de 0.8.

El refuerzo para fuerza cortante debe estar formado por estribos verticales cerrados de una pieza, de diámetro no menor de 7.9 mm (No 2.5), rematados como se indica en el inciso 3.5.1.2.3.

3.5.1.3 Miembros a flexocompresión

Los requisitos de esta sección se aplican a miembros en los que la carga axial de diseño sea tal que.

$$P_u > 0.1 A_g f'_c \tag{3.4}$$

3.5.1.3.1 Requisitos geométricos

a) La dimensión transversal mínima no debe ser menor que 30 cm.
b) El área A_g, debe cumplir con la condición.

$$A_g \geq \frac{P_u}{0.5 f'_c} \tag{3.5}$$

c) La relación entre la menor dimensión transversal y la dimensión transversal perpendicular no debe ser menor que 0.4.
d) La relación entre la altura libre y la menor dimensión transversal no debe exceder de 15.

3.5.1.3.2 Resistencia mínima a flexión

Las resistencias a flexión de las columnas en un nudo deben satisfacer la siguiente condición.

$$\sum M_e \geq 1.5 \sum M_g \tag{3.6}$$

donde.

$\sum M_e$ = Suma de los momentos resistentes de diseño de las columnas que llegan a ese nudo, referidas al centro del nudo

$\sum M_g$ = Suma de los momentos resistentes de diseño de las vigas que llegan al nudo, referidas al centro de éste

Las sumas anteriores deben realizarse de modo que los momentos de las columnas se opongan a los de las vigas. La condición debe cumplirse para los dos sentidos en que puede actuar el sismo.

Al calcular la carga axial de diseño para la cual se valúe el momento resistente, M_e, de una columna, la fracción de dicha carga debida al sismo se debe tomar igual al doble de la calculada, cuando esto conduzca a un momento resistente menor. En tal caso, la columna se debe dimensionar al tomar en cuenta el incremento de carga mencionda. El factor de resistencia por flexocompresión se debe tomar igual a 0.8.

Como opción, las columnas pueden dimensionarse con los momentos y fuerzas axiales de diseño obtenidos del análisis, si el factor de resistencia por flexocompresión se le asigna el valor de 0.6.

3.5.1.3.3 Refuerzo longitudinal

La cuantía del refuerzo longitudinal, p, debe satisfacer la siguiente condición.

$$0.01 \leq p \leq 0.04 \qquad (3.7)$$

Solo se deben formar paquetes de dos barras.

El traslapa de barras longitudinales solo se permite en la mitad del elemento; estos traslapes deben cumplir con los requisitos del inciso 3.9 (NTC para diseño y construcion de estructuras de concreto del RCDF87). Las uniones soldadas o con dispositivos mecánicos que cumplan con los requisitos del inciso 3.9 (NTC para diseño y construcion de estructuras de concreto del RCDF87), pueden usarse en cualquier localización con tal de que en una misma sección cuando mas se unan barras alternadas y que las uniones de barras adyacentes no disten entre si menos de 60 cm en la dirección longitudinal del miembro.

El refuerzo longitudinal debe cumplir con las disposiciones del inciso 3 (NTC para diseño y construcion de estructuras de concreto del RCDF87) que no se modifican en este inciso.

3.5.1.3.4 Refuerzo transversal

Debe cumplirse con los requisitos del inciso 3.3 (NTC para diseño y construcion de estructuras de concreto del RCDF87) y los del inciso siguiente (inciso 3.5.1.3.5), y con los requisitos mínimos que aquí se establecen. No debe ser de grado mayor que el 42.

Se debe suministrar el refuerzo transversal mínimo que se especifica enseguida en ambos extremos de la columna, en una longitud no menor que.

a) La mayor dimensión transversal de ésta
b) Un sexto de su altura libre
c) 60 cm

En la parte inferior de columnas de planta baja este refuerzo debe llegar hasta media altura de la columna, y debe continuarse dentro de la cimentación, al menos, una distancia igual a la longitud de desarrollo en compresión de la barra mas gruesa (en los nudos se debe cumplir con los requisitos del inciso 3.5.1.4 que se indican posteriormente.

a) En columnas de núcleo circular, la cuantía volumétrica de refuerzo helicoidal o de estribos circulares, P_s, debe cumplir con la siguiente relación.

$$p_s \geq 0.45\left(\frac{A_g}{A_c} - 1\right)\frac{f'_c}{f_y}$$

$$p_s \geq 0.12\frac{f'_c}{f_y} \qquad (3.8)$$

b) En columnas de núcleo rectangular, la suma de las áreas de estribos y grapas, A_{sh}, en cada dirección de la sección de la columna debe cumplir con la relación.

$$A_{sh} \geq 0.3\left(\frac{A_g}{A_c} - 1\right)\frac{f'_c}{f_y}sh_c$$

$$A_{sh} \geq 0.12\frac{f'_c}{f_y}sh_c \qquad (3.9)$$

donde:

A_c = Area tranvsversal del nucleo,hasta la orilla exterior del refuerzo transversal

A_c = Area tranvsversal de la columna

f_y = Esfuerzo de fluencia del refuerzo transversal

(3.10) h_c = Dimensión del núcleo, normal al refuerzo de área A_{sh}

s = Separación del refuerzo transversal

Este refuerzo transversal debe estar formado por estribos de una pieza, sencillos o sobrepuestos, de diámetro no menor que 9.5 mm (No 3) y rematados como se indica en el inciso 3.5.1.2.3. Puede complementarse con grapas del mismo diámetro que los estribos, espaciados igual que éstos a lo largo del miembro. Cada extremo de una grapa debe abrazar a una barra longitudinal de la periferia con doblez de 135 grados, seguido de un tramo recto de, al menos, 10 diámetros de la grapa.

La separación del refuerzo transversal no debe exceder de la cuarta parte de la menor dimensión transversal del elemento, ni de 10 cm.

La distancia centro a centro, transversal al eje del miembro, enre ramas de estribos sobrepuestos no debe ser mayor de 45 cm, y entre grapas y ramas de estribos sobrepuestos no debe ser mayor de 25 cm. Si el refuerzo consta de estribos sencillos, la mayor dimensión de éstos no debe exceder de 45 cm.

En el resto de la columna el refuerzo transversal debe cumplir con los requisitos del inciso 3 (NTC para diseño y construcion de estructuras de concreto del RCDF87).

3.5.1.3.5 Requisitos para fuerza cortante

Los elementos a flexocompresión se deben dimensionar de manera que no fallen por fuerza cortante antes que se formen las articulaciones plásticas en las vigas. Para ello la fuerza cortante de diseño se debe obtener del equilibrio del elemento en su altura libre al suponer en cada extremo un momento igual a la mitad de $1.5\Sigma M_g$ (definida en la sección 3.5.1.3.2. En el extremo inferior de columnas de planta baja se debe usar el momento resistente de diseño de la columna obtenido con la carga axial de diseño que conduzca al mayor momento resistente. En el extremo superior de columnas del último entrepiso se debe usar $1.5\Sigma M_g$.

Cuando las columnas se dimensionen por flexocompresión con el procedimiento optativo incluído en el inciso 3.5.1.3.2, el dimensionamiento por fuerza cortante se debe realizar a partir de la fuerza de diseño obtenida del análisis, y utilizar un factor de resistencia igual a 0.5.

En elementos a flexocompresión en que la fuerza axial de diseño, incluyendo los efectos del sismo, sea menor que $A_g f'_c/20$, al calcular el refuerzo para fuerza cortante, si la fuerza cortante de diseño causada por el sismo es igual o mayor que la mitad de la fuerza cortante de diseño calculada según los párrafos anteriores, se puede despreciar la contribución del concreto V_c.

El refuerzo para fuerza cortante debe estar formaddo por estribos cerados, de una pieza, rematados como se indica en el inciso 3.5.1.2.3, o por hélices continuas, ambos de diámetros no menor que 9.5 mm (No 3) y de grado no mayor que el 42.

3.5.1.4 Uniones viga-columna

3.5.1.4.1 Requisitos generales

Las fuerzas que intervienen en el dimensionamiento por fuerza cortante de la unión se deben determinar al suponer que el esfuerzo de tensión en las barras longitudinales de las vigas que llegan a la unión es 1.25 f_y.

El refuerzo longitudinal de las vigas que llegan a la unión debe pasar dentro del núcleo de la columna.

En los planos estructurales deben incluírse dibujos, acotados y a escala, del refuerzo en las uniones viga-columna.

Una unión viga-columna o nudo se define como aquella parte de la columna comprendida en el peralte de las vigas que llegan a ella.

3.5.1.4.2 Refuerzo transversal

En un nudo debe suministrarse el refuerzo transversal mínimo especificado en el inciso 3.5.1.3.4. Si el nudo está confinado por cuatro trabes que llegan a él y el ancho de cada una es, al menos, igual a 0.75 veces el ancho respectivo de la columna, puede usarse la mitad del refuerzo transversal mínimo.

3.5.1.4.3 Resistencia a fuerza cortante

Se debe admitir revisar la resistencia del nudo a fuerza cortante en cada dirección principal de la sección en forma independiente. La fuerza cortante se debe calcular en un plano horizontal a media altura del nudo.

En nudos confinados como se dice en el inciso 3.5.1.4.2, la resistencia de diseño a fuerza cortante se debe tomar igual a

$$5.5 F_R \sqrt{f_c^*}\, b_e h \qquad (3.11)$$

En otros nudos se debe tomar igual a.

$$4.5 F_R \sqrt{f_c^*}\, b_e h \qquad (3.12)$$

b_e es el ancho efectivo del nudo
h es la dimensión transversal de la columna en la dirección de la fuerza.

El ancho b_e debe tomar igual al promedio del ancho de la o las vigas consideradas y la dimensión transversal de la columna normal a la fuerza, pero no mayor que el ancho de la o las vigas mas h.

3.5.1.4.4 Anclaje del refuerzo

Toda barra de refuerzo longitudinal de vigas que termine en un nudo debe prolongarse hasta la cara lejana del núcleo de la columna y rematarse con un doblez a 90 grados, seguido de un tramo recto no menor de 12 diámetros. La sección crítica para revisar el anclaje de estas barras debe ser el plano externo del núcleo de la columna. La revisión se debe efectuar de acuerdo con la sección 3.1.1c (NTC para diseño y construcion de estructuras de concreto del RCDF87), donde es suficiente usar una longitud de desarrollo del 90 por ciento de la allí determinada.

Los diámetros de las barras de vigas y columnas que pasen rectos a través de un nudo deben seleccionarse de modo que cumplan las relaciones siguientes:

$$h(columna)/d_b(barras\ de\ viga) \geq 20$$
$$h(viga)/d_b(barras\ de\ columna) \geq 20 \qquad (3.13)$$

donde h(columna) es la dimensión transversal de la columna en la dirección de las barras de viga consideradas.

Si en la columna superior del nudo se cumple que:

$$\frac{P_u}{A_g f'_c} \geq 0.3 \qquad (3.14)$$

se puede tomar la relación siguiente:

$$h(viga)/d_b(barras\ de\ columna) \geq 15 \qquad (3.15)$$

La relación dada por la Ec 3.15 también es suficiente cuando en la estructura los muros de concreto reforzado resisten más del 50 por ciento de la fuerza lateral total.

3.5.1.5 Sistemas losa plana-columnas para resistir sismo

Si la altura de la estructura no excede de 20 m y, además, existen al menos tres crujías en cada dirección o jay trabes de borde, para el diseño por sismo se puede usar Q=3; también puede aplicarse este valor cuando el sistema se combine con muros de concreto reforzado que cumplan con 4.5.2, incluyendo el inciso b de esa sección (NTC para diseño y construcion de estructuras de concreto del RCDF87), y que resistan no menos del 75 por ciento de la fuerza lateral. Cuando no se satisfagan las condiciones anteriores, se debe usar Q=2. Con relación a los valores de Q, debe cumplirse, además, con los correspondientes incisos anteriores (que es el Cap 5 de las NTC para diseño y construcion de estructuras de concreto del RCDF87). En todos los casos se deben respetar las disposiciones siguientes:

I Las columnas deben cumplir con los requisitos de 3.5.1.3 para columnas de marcos dúctiles, excepto en lo referente al dimensionamiento por flexocompresión, el cual sólo se debe realizar mediante el procedimiento optativo que se establece en el inciso 3.5.1.3.2.

II Las uniones losa-columna deben cumplir con los requisitos de 3.5.1.4 para uniones viga-columna, con las salvedades que siguen:

No es necesaria la revisión de la resistencia del nudo a fuerza cortante, sino basta cumplir con el refuerzo transversal prescrito en 3.5.1.4.2 para nudos confinados.

Los requisitos de anclaje de 3.5.1.4.4 se deben aplicar al refuerzo de la losa que pase por el núcleo de una columna. Los diámetros de las barras de la losa y columnas que pasen rectas a través de un nudo deben seleccionarse de modo que se cumplan las relaciones siguientes:

$$\begin{aligned} h(columna)/d_b(barras\ de\ losa) &\geq 20 \\ h(losa)/d_b(barras\ de\ columna) &\geq 15 \end{aligned} \qquad (3.16)$$

donde h(columna) es la dimensión transversal de la columna en la dirección de las barras de losa consideradas.

3.5.2 Condiciones para estructuras dúctiles de acero

Con base en los puntos I.4 y II del inciso 3.5, se reproduce el Capítulo 11, Estructuras dúctiles, de las NTC para diseño y construción de estructuras metálicas del RCDF87.

3.5.2.1 Alcance

En este capítulo se indican los requisitos que deben cumplirse para que puedan adoptarse valores del factor de comportamiento sísmico Q iguales a 4.0 o 3.0.

3.5.2.2 Marcos dúctiles

3.5.2.2.1 Requisitos generales

Se indican aquí los requisitos que debe satisfacer un marco rígido de acero estructural para ser considerado un marco dúctil. Estos requisitos se aplican a marcos rígidos diseñados con un factor de comportamiento sísmico Q igual a 4.0 o a 3.0, que formen parte de sistemas estructurales que cumplan las condiciones enunciadas en el capítulo 5, partes I y II, de las NTC para diseño por sismo, necesarias para utilizar ese valor del factor de comportamiento sísmico.

Tanto en los casos en que la estructura está formada sólo por marcos como por aquellos en que está compuesta por marcos y muros o contravientos, cada uno de los marcos se debe diseñar para resistir, como mínimo, fuerzas horizontales iguales al 25 por ciento de las que le corresponderían si trabajase aislado del resto de la estructura.

La gráfica esfuerzo de tensión-deformación del acero empleado debe tener una zona de cedencia, de deformación creciente bajo esfuerzo prácticamente constante, correspondiente a un alargamiento máximo no menor de uno por ciento, seguida de un endurecimiento por deformación. El alargamiento correspondiente a la ruptura no debe ser menor de 20 por ciento.

3.5.2.2.2 Miembros en flexión

Los requisitos de esta sección se aplican a miembros principales que trabajan esencialmente en flexión. Se incluyen vigas y columnas con cargas axiales pequeñas, tales que P_u no exceda de $P_y/10$.

3.5.2.2.2.1 Requisitos geométricos

Todas las vigas deben ser de sección transversal I o rectangular hueca, excepto en los casos cubiertos en el inciso 3.5.2.2.5.

El claro libre de las vigas no debe ser menor que cinco veces el peralte de su sección transversal, ni el ancho de sus patines mayor que el ancho del patín o el peralte del alma de la columna con la que se conecten.

El eje de las vigas no debe separarse horizontalmente del eje de las columnas más de un décimo de la dimensión transversal de la columna normal a la viga.

Las secciones transversales de las vigas deben ser tipo 1, de manera que han de satisfacer los requisitos geométricos que se indican en los incisos 2.3.1 y 2.3.2 (NTC para diseño y construción de estructuras metálicas del RCDF87). Sin embargo, se permite que la relación ancho/grueso del alma llegue hasta $5300/\sqrt{F_y}$ si en las zonas de formación de articulaciones plásticas se toman las medidas necesarias (refuerzo del alma mediante atiesadores transversales o placas adosadas a ella, soldadas adecuadamente) para impedir que el pandeo local se presente antes de la formación del mecanismo de colapso.

Además, las secciones transversales deben tenes dos ejes de simetría, una vertical, en el plano en que actúan las cargas gravitacionales, y otro horizontal. Cuando se utilicen cubreplacas en los patines para aumentar la resistencia del perfil, deben conservarse los dos ejes de simetría.

Si las vigas están formadas por placas soldadas, la soldadura entre almas y patines debe ser continua en toda la longitud de la viga, y en las zonas de formación de articulaciones plásticas debe ser capaz de desarrollar la resistencia total en cortante de las almas.

Cuando se empleen vigas de resistencia variable, ya sea por adición de cubreplacas en algunas zonas o porque su peralte varíe a lo largo del claro, el momento resistente nunca debe ser menor, en ninguna sección, que la cuarta parte del momento resistente máximo, que se tendrá en los extremos.

En estructuras soldadas deben evitarse los agujeros, siempre que sea posible, en ls zonas de formación de articulaciones plásticas. En estructuras atornilladas o remachadas, los agujeros que sean necesarios en la parte del perfil que trabaje en tensión se deben punzar a un diámetro menor y se agrandan después, hasta darles el diámetro completo, con un taladro o un escarificador. este mismo procedimiento se debe seguir en estructuras soldadas, si se requieren agujeros para montaje o con algún otro objeto. Para los fines de los dos párrafos anteriores, las zonas de formación de articulaciones plásticas se consideran de longitud igual a un peralte, en los extremos de las vigas, y a dos peraltes, medido s uno a cada lado de la sección en la que aparece, en teoría, la articulación plástica, en zonas intermedias.

En aceros cuyo esfuerzo mínimo especificado de ruptura en tensión, F_u, es menor 1.5 veces el esfuerzo de fluencia mínimo garantizado, F_y, no se debe permitir la formación de articulaciones plásticas en zonas en que se haya reducido el área de los patines, ya sea por agujeros para tornillos o por cualquier otra causa.

No se deben hacer empalmes de ningún tipo, en las vigas propiamente dicha o en sus cubreplacas, en zonas de formación de articulaciones plásticas.

3.5.2.2.2.2 Requisitos para fuerza cortante

Los elementos que trabajan principalmente en flexión se deben dimensionar de manera que no se presenten fallas por cortante antes de que se formen las articulaciones plásticas asociadas con el mecanismo de colapso. Para ello, la fuerza cortante de diseño se obtiene del equilibrio del miembro entre las secciones en que se forman las articulaciones plásticas, en los que se supone que actúan momentos del mismo sentido y de magnitudes iguales a los momentos plásticos resistentes del elemento en esas secciones, sin factores de redución, y evaluados al tomar el esfuerzo de fluencia del material igual a 1.25 F_y. Al plantear la ecuación de equilibrio para calcular la fuerza cortante se deben tener en cuenta las cargas transversales que obran sobre el miembro, multiplicadas por el factor de carga.

Como una opción se permite hacer el dimensionamiento al tomar como base las fuerzas cortantes de diseño obtenidas en el análisis, pero utilizar un factor de resistencia F_R igual a 0.7, en lugar del valor de 0.9 especificado en el artículo 3.3.3 (NTC para diseño y construción de estructuras metálicas del RCDF87).

Las articulaciones plásticas se forman, en la mayoría de los casos, en los extremos de los elementos que trabajan en flexión. Sin embargo, hay ocasiones frecuentes en las vigas de los niveles superiores de los edificios, en que una de ellas se forma en la zona central del miembro. Cuando esto suceda, la fuerza cortante debe evaluarse al tener en cuenta la posición real de la articulación plástica.

3.5.2.2.2.3 Contraventeo lateral

Deben soportarse lateralmente todas las secciones transversales de las vigas en las que puedan formarse articulaciones plásstica asociadas con el mecanismo de colapso. Además, la distancia entre cada una de estas secciones y la siguiente sección soportada lateralmente no debe ser mayor que la dada a continuación.

$$L_p = 1250\frac{r_y}{\sqrt{F_y}} \qquad (3.17)$$

Este requisito se aplica a un solo lado de la articulación plástica cuando ésta se forma en un extremo de la viga, y en ambos lados cuando aparece en una sección intermedia. La expresión anterior es válida para vigas de sección transversal I o H, flexionadas alrededor de su eje de mayor momento de inercia.

En zonas que se conservan en el intervalo elástico al formarse el mecanismo de colapso, la separación entre puntos no soportados lateralmente puede ser mayor que la indicada en el párrafo anterior, pero no debe esceder el valor de L_u , calculado de acuerdo con el inciso 3.3.2.2 (NTC para diseño y construción de estructuras metálicas del RCDF87).

Los elementos de contraventeo deben proporcionar soporte lateral, directo o indirecto, a los dos patines de las vigas. Cuando el sistema de piso proporcione soporte lateral al patín superior, el desplazamiento lateral del patín inferior puede evitarse por medio de atiesadores verticales de rigidez adecuada, soldados a los dos patines y al alma de la viga.

3.5.2.2.3 Miembros en flexocompresión

Los requisitos de esta sección se aplican a miembros que trbajan en flexocompresión, en los que la carga axial de diseño, P_u, es mayor que $P_u/10$. La mayoría de estos miembros son columnas, pero pueden ser de algún otro tipo; por ejemplo, vigas que forman parte de crujías contraventeadas de marcos rígidos han de diseñarse, en general, como elementos flexocomprimidos.

3.5.2.2.3.1 Requisitos geométricos

Si la Sección transversal es rectangular hueca, la relación de la mayor a la menor de sus dimensiones exteriores no debe exceder de 2 y la dimensión menor debe ser mayor o igual a 20 cm.

Si la sección transversal es H, el ancho de los patines no debe ser mayor que el peralte total, la relación peralte-ancho del patín no debe exceder de 1.5, y el ancho de los patines debe ser mayor o igual a 20 cm.

La relación de esbeltez máxima de las columnas no debe exceder de 60.

3.5.2.2.3.2 Resistencia mínima en flexión

La resistencia en flexión de las columnas que concurren a un nudo deben satisfacer la condición dada por la Ec 5.8.5 del inciso 5.8.5 (NTC para diseño y construción de estructuras metálicas del RCDF87), con las excepciones que se indican en este inciso.

Como una opción, se permite hacer el dimensionamiento al tomar como base los elementos mecánicos de diseño obtenidos en el análisis, y reducir el factor de resistencia F_R utilizado en flexocompresión de 0.9 a 0.7.

3.5.2.2.3.3 Requisitos para fuerza cortante

Los elementos flexocomprimidos se deben dimensionar de manera que no fallen prematuramente por fuerza cortante. Para ello, la fuerza cortante de diseño se otiene del equilibrio del miembro, al considerar su longitud igual a la altura libre y suponer que en sus extremos obran momentos del mismo sentido y de magnitud igual a los momentos máximos resistentes de las columnas en el plano de estudio, que valen $Z_c(F_{yc}-f_a)$. El significado de las literales que aparecen en esta expresión se explica con referencia a la Ec 5.8.5 del inciso 5.8.5 (NTC para diseño y construción de estructuras metálicas del RCDF87).

Cuando las columnas se dimensionen por flexocompresión con el procedimiento optativo del inciso 3.5.2.2.3.2, la revisión por fuerza cortante se debe realizar con la fuerza de diseño obtenida en el análisis y utilizar un factor de resistencia de 0.7.

3.5.2.2.4 Uniones viga-columna

Las uniones viga-columna deben satisfacer las recomendaciones de la sección 5.8 "Conexiones rígidas entre vigas y columnas" (NTC para diseño y construción de estructuras metálicas del RCDF87), con las modificaciones pertinentes cuando las columnas sean de sección transversal rectangular hueca.

3.5.2.2.4.1 Contraventeo

Si en alguna junta de un marco dúctil no llegan vigas al alma de la columna, por ningún lado de ésta, o si el peralte de la viga o vigas que llegan por alma es apreciablemente menor que el de las que se apoyan en los patines de la columna, éstos deben ser soportados lateralmente al nivel de los patines inferiores de las vigas.

3.5.2.2.4.2 Vigas de alma abierta (armaduras)

En esta sección se indican los requisitos especiales que deben satisfacerse cuando se desea emplear vigas de alma abierta (armaduras) en marcos dúctiles. Deben cumplirse, además, todas las condiciones aplicables de este capítulo.

Las armaduras pueden utilizarse como miembros horizontales en marcos dúctiles, si se diseñan de manera que la suma de las resistencias en flexión ante fuerzas sísmicas de las dos armaduras que concurran en cada nudo intermedio sea igual o mayor 1.25 veces la suma de las resistencias en flexión ante fuerzas sísmicas de ls columns que llegan al nudo. En nudos extremos, el requisito anterior debe ser satisfecho por la única armadura que forma parte de ellos.

Además, deben cumplirse las condiciones siguientes:

a) Los elementos de las armaduras que trabajan en compresión o en flexocompresión, sean cuerdas, diagonales o montantes, se deben diseñar con un factor de resistencia, F_R, igual a 0.7. Al determinar cuales elementos trabajan en compresión o en flexocompresión deben tomarse en cuenta los dos sentidos en que actúa el sismo de diseño.

b) Las conexiones entre las cuerdas de las armaduras y las columnas deben ser capaces de desarrollar la resistencia correspondiente al flujo plástico de las cuerdas.

c) En edificios de más de un piso, el esfuerzo en las columnas producido por las fuerzas axiales de diseño no deben ser mayores de 0.30 F_y, y la relación de esbeltez máxima de ls columnas no debe exceder de 60.

3.6 Espectros para diseño sísmico

De acuerdo con las NTC para diseño por sismo, cuando se aplique el análisis dinámico modal que especifica la sección 9 de sus normas, se adoptan las siguientes hipótesis para el análisis de la estructura:

La ordenada del espectro de aceleraciones para diseño sísmico, **a**, expresada como fracción de la aceleración de la gravedad, está dada por las siguientes expresiones:

$$
\begin{aligned}
a &= \frac{1}{4}\left(1 + 3\frac{T}{T_a}\right)c & \forall\, T < T_a \\
a &= c & \forall\, T_a \le T \le T_b \\
a &= \left(\frac{T_b}{T}\right)^r c & \forall\, T > T_b
\end{aligned}
\tag{3.18}
$$

T es el período natural de interés; T, T_a, y T_b están expresados en segundos; c es el coeficiente sísmico, y r un exponente que depende de la zona en que se halla la estructura, y se expecifica en la tabla 3.1 de las NTC para diseño por sismo, reproducida a continuación.

El coeficiente sísmico c se obtiene del Art 206 del RCDF87, salvo que la parte sombreada de la zona II de la fig 3.1 de las NTC para diseño por sismo (NTC-sismo) se debe tomar c = 0.4 para las estructuras del grupo B, y c = 0.6 para las del A.

Tabla 3.1 Valores de c, T_a, T_b, y r

Zona	c	T_a(s)	T_b(s)	r
I	0.16	0.2	0.6	1/2
II*	0.32	0.3	1.5	2/3
III+	0.40	0.6	3.9	1

Notas: Coeficiente sísmico para construcciones del Grupo B
*** No sombreada (Fig 3.1, NTC-sismo)**
+ Y parte sombreada de zona II (Fig 3.1, NTC-sismo)

4. FUERZAS SISMICAS

En este capítulo se describen los métodos que considera el RCDF87 para cuantificar las fuerzas que se deben considerar en el diseño de una edificación para soportar los efectos de un sismo.

4.1 Anális dinámico

De acuerdo con las NTC para diseño por sismo, toda estructura puede analizarse mediante un método dinámico. Se aceptan como métodos de análisis dinámico:

a) El modal (modal espectral)

b) El paso a paso de respuestas a sismos específicos

A fin de explicar los métodos para analizar las estructuras ante cargas dinámicas, se presentan los siguientes desarrollos:

4.1.1 Ecuaciones de equilibrio dinámico de las edificaciones

Las ecuaciones de equilibrio dinámico de los modelos estructurales lineales para edificaciones se pueden expresar como:

$$\tilde{M}\frac{d^2}{dt^2}\vec{u}(t) + \tilde{C}\frac{d}{dt}\vec{u}(t) + \tilde{K}\vec{u}(t) = \vec{F}(t) \qquad (4.1)$$

Con las siguientes condiciones iniciales

$$\begin{aligned}
\frac{d}{dt}\vec{u}(t)\big|_{t=0} &= \vec{v}_0 \\
&= \text{vector de velocidades conocido} \qquad (4.2)\\
\vec{u}(t)\big|_{t=0} &= \vec{u}_0 \\
&= \text{vector de desplazamientos conocido}
\end{aligned}$$

donde, para la edificación en particular, se definen los siguientes conceptos.

$$\begin{aligned}
\vec{M} &= \text{Matriz de masas} \\
\vec{C} &= \text{Matriz de amortiguamientos} \\
\vec{K} &= \text{Matriz de rigideces} \\
\vec{u}(t) &= \text{vector de desplazamientos} \\
\frac{d}{dt}\vec{u}(t) &= \text{vector de velocidades} \qquad (4.3)\\
\frac{d^2}{dt^2}\vec{u}(t) &= \text{vector de aceleraciones} \\
\vec{F}(t) &= \text{vector de cargas}
\end{aligned}$$

En el caso de fuerzas sismicas, el vector de cargas se puede expresar en términos del vector de aceleraciones del terreno (acelerograma), $\ddot{u}_g(t)$,de acuerdo con la expresión siguiente:

$$\vec{F} = -\vec{M}\vec{1}\ddot{u}_g(t) \qquad (4.4)$$

donde

$$\begin{aligned}
\vec{1}^T &= [1\ 1\ .\ .\ .\ 1] \\
&= \text{vector con componentes unitarias} \qquad (4.5)
\end{aligned}$$

4.1.2 Métodos directos de integración paso a paso

Los métodos que actualmente se utilizan para integrar paso a paso las ecuaciones de quilibrio dinámico de las edificaciones se agrupan en:

a) métodos directos

b) métodos de superposición modal

El método directo que mas se utiliza es el denominado método de Newmark. Este método se basa en la aproximación lineal de la aceleración en el tamaño del paso de integración, según se muestra en la Fig 4.1.

De acuerdo con la hipótesis de la aceleración lineal, los elementos de las ecuaciones de equilibrio dinámico (Ec 4.1) al final del paso de integración se pueden escribir como.

$$
\begin{aligned}
\frac{d^2}{dt^2}\vec{u}_{t+\Delta t} &= \frac{d^2}{dt^2}\vec{u}_{t+\Delta t} \\
\frac{d}{dt}\vec{u}_{t+\Delta t} &= \frac{d}{dt}\vec{u}_t + \frac{1}{2}\Delta t\left(\frac{d^2}{dt^2}\vec{u}_{t+\Delta t} + \frac{d^2}{dt^2}\vec{u}_t\right) \\
\vec{u}_{t+\Delta t} &= \vec{u}_t + \Delta t\frac{d}{dt}\vec{u}_t + \frac{1}{6}(\Delta t)^2\left(\frac{d}{dt}\vec{u}_{t+\Delta t} + 2\frac{d}{dt}\vec{u}_t\right)
\end{aligned}
\tag{4.6}
$$

La aproximación de Newmark consiste en:

$$
\begin{aligned}
\frac{d}{dt}\vec{u}_{t+\Delta t} &= \frac{d}{dt}\vec{u}_t + (1-\gamma)\Delta t\frac{d^2}{dt^2}\vec{u}_t + \gamma\Delta t\frac{d^2}{dt^2}\vec{u}_{t+\Delta t} \\
&= \vec{a} + \gamma\Delta t\frac{d^2}{dt^2}\vec{u}_{t+\Delta t} \\
\vec{u}_{t+\Delta t} &= \vec{u}_t + \Delta t\frac{d}{dt}\vec{u}_t + \left(\frac{1}{2} - \beta\right)(\Delta t)^2\frac{d^2}{dt^2}\vec{u}_t + \beta(\Delta t)^2\frac{d^2}{dt^2}\vec{u}_{t+\Delta t} \\
&= \vec{b} + \beta(\Delta t)^2\frac{d^2}{dt^2}\vec{u}_{t+\Delta t}
\end{aligned}
\tag{4.7}
$$

donde:

$$\vec{a} = \frac{d}{dt}\vec{u}_t + (1 - \gamma)\Delta t \frac{d^2}{dt^2}\vec{u}_t$$

$$\vec{b} = \vec{u}_t + \Delta t \frac{d}{dt}\vec{u}_t + (\frac{1}{2} - \beta)(\Delta t)^2 \frac{d^2}{dt^2}\vec{u}_t \tag{4.8}$$

El parámetro β está relacionado con la estabilidad del método (para β = 1/4, el método es incondicionalmente estable) y el parámetro se relaciona con la estabilidad y convergencia del método debido al amortiguamiento matemático que puede inducirse (para = 1/2, no se presenta el amortiguamiento matemático).Para el caso en que = 1/6 y = 1/2, las Ec 4.7 se reducen a las correspondientes Ec 4.6.

Al valuar las ecuaciones de equilibrio dinámico (Ec 4.1) al final del paso de integración (en t = t + t) y al sustituír en la ecuación resultante a las Ec 4.7 se obtiene la siguiente ecuación.

$$\vec{M}\frac{d^2}{dt^2}\vec{u}_{t+\Delta t} + \vec{C}\left(\vec{a} + \gamma\Delta t \frac{d^2}{dt^2}\vec{u}_{t+\Delta t}\right) + \vec{K}\left(\vec{b} + \beta(\Delta t)^2 \frac{d^2}{dt^2}u_{t+\Delta t}\right) = \vec{F}_{t+\Delta t} \tag{4.9}$$

La Ec 4.9 puede escribirse como:

$$\vec{K}^* \frac{d^2}{dt^2}\vec{u}_{t+\Delta t} = \vec{p} \tag{4.10}$$

donde:

$$\vec{K}^* = \vec{M} + \gamma\Delta t\vec{C} + \beta(\Delta t)^2\vec{K}$$

$$\vec{p} = \vec{F}_{t+\Delta t} - \vec{C}\vec{a} - \vec{K}\vec{b} \tag{4.11}$$

La Ec 4.10 permite cuantificar la aceleración al final del paso es un sistema de ecuaciones algebraicas lineales, simétricas, de coeficientes constantes si el paso de integración se conserva constante durante el proceso de integración.

En la dinámicaestructural se acostumbra cuantificar a la matriz de amortiguamientos de la estructura de acuerdo con el criterio de Rayleigh, expresado mediante la siguiente ecuación.

$$\vec{C} = \alpha\vec{M} + \mu\vec{K} \tag{4.12}$$

Al sustituír la Ec 4.12 en las Ec 4.11 se obtiene.

$$\begin{aligned} \tilde{K}^{*} &= (1 + \alpha\gamma\Delta t)\tilde{M} + (\gamma\mu\Delta t + \beta(\Delta t)^{2})\tilde{K} \\ \vec{p} &= \vec{F}_{t+\Delta t} - \alpha\tilde{M}\vec{a} - \tilde{K}(\mu\vec{a} + \vec{b}) \end{aligned} \tag{4.13}$$

El algoritmo del método de integración paso a paso de Newmark, resumido por las Ec 4.10 y 4.13, necesariamente se debe llevar a cabo en una computadora debido al número de operaciones que involucra.

4.1.3 Método directo paso a paso de superposición modal

Otra forma de integrar paso a paso las ecuaciones de equilibrio dinámico de las estructuras (Ec 4.1) es mediante la solución del problema de eigenvalores, según se indica a continuación.

4.1.3.1 Solución del problema de valores característicos (eigenvalores) de las ecuaciones de equilibrio dinámico

Este caso corresponde a un problema de vibraciones libres no amortiguadas, cuyas ecuaciones resultan ser.

$$\tilde{M}\frac{d^2}{dt^2}\vec{u}(t) + \tilde{K}\vec{u}(t) = \vec{0} \tag{4.14}$$

En las vibraciones libres el movimiento es armónico, es decir.

$$\frac{d^2}{dt^2}\vec{u}(t) = -\omega^2\vec{u}(t) \tag{4.15}$$

y las ecuaciones de vibración libre resultan ser

$$\tilde{K}\vec{u} = \omega^2\tilde{M}\vec{u} \tag{4.16}$$

que es el clásico problema de eigenvalores comunmente expresado como:

$$\tilde{A}\vec{x} = \lambda\tilde{B}\vec{x} \tag{4.17}$$

Varios son los métodos que existen para resolver el problema de eigenvalores. Los utilizados con las computadoras, entre otros, se pueden nombrar a

. El de Jacobi

. El de la iteración del subespacio

Cuando se emplean calculadoras de escritorio para los modelos extructurales mas simples (rigideces de entrepiso y masas con movimientos unidireccionales) se utilizan los métodos de:

. Stodolla-Vianelo-Newmark

. Holzer

4.1.3.2 Desacoplamiento de las ecuaciones de equilibrio dinámico

La transformación que permite desacoplar las ecuaciones de equilibrio dinámico se puede expresar como.

$$\vec{u} = \vec{\vec{R}}\vec{y} \tag{4.18}$$

donde

$$\begin{aligned} \vec{y} &= \text{vector del nuevo sistema coordenado} \\ \vec{\vec{R}} &= [\vec{r}^1 \ \vec{r}^2 \ \vec{r}^3 \ \ldots \ \vec{r}^n] \\ &= \text{Matriz modal} \\ \vec{r}^n &= \text{n-ésimo eigenvector} \end{aligned} \tag{4.19}$$

De acuerdo con la transformación de coordenadas anterior (Ec 4.18) las expresiones de los vectores de velocidad y de acelaración resultan ser:

$$\begin{aligned} \frac{d}{dt}\vec{u}(t) &= \vec{\vec{R}}\frac{d}{dt}\vec{y}(t) \\ \frac{d^2}{dt^2}\vec{u}(t) &= \vec{\vec{R}}\frac{d^2}{dt^2}\vec{y}(t) \end{aligned} \tag{4.20}$$

De acuerdo con las Ec 4.18 y 4.20 las ecuaciones de equilibrio dinámico (Ec 4.1) en el sistema de referencia transformado se expresan como:

$$\vec{M}\vec{R}\frac{d^2}{dt^2}\vec{y}(t) + \vec{C}\vec{R}\frac{d}{dt}\vec{y}(t) + \vec{K}Rdyd\vec{y}(t) = \vec{F}(t) \qquad (4.21)$$

Al premultiplicar la Ec 4.21 por la transpuesta de la matriz modal se obtiene la siguiente expresión.

$$\vec{R}^T\vec{M}\vec{R}\frac{d^2}{dt^2}\vec{y}(t) + \vec{R}^T\vec{C}\vec{R}\frac{d}{dt}\vec{y}(t) + \vec{R}^T\vec{K}Rdyd\vec{y}(t) = \vec{R}^T\vec{F}(t) \qquad (4.22)$$

Al definir los siguientes conceptos

$$\begin{aligned}
\vec{M}^* &= \vec{R}^T\vec{M}\vec{R} = \text{Matriz de masas transformada} \\
\vec{C}^* &= \vec{R}^T\vec{C}\vec{R} = \text{Matriz de amortiguamientos transformada} \\
\vec{K}^* &= \vec{R}^T\vec{K}\vec{R} = \text{Matriz de rigideces transformada} \\
\vec{F}^*(t) &= \vec{R}^T\vec{F}(t) = \text{vector de cargas transformado}
\end{aligned} \qquad (4.23)$$

De acuerdo con las propiedades de ortogonalidad de los eigenvectores respecto a las matrices de masas y de rigideces, la matriz de masas transformada y la matriz de rigideces transformada resultan ser matrices diagonales. Si la matriz de amortiguamientos se selecciona de tal manera que también la matriz de amortiguamientos transformada sea una matriz diagonal, las ecuaciones de equilibrio dinámico transformadas (Ec 4.22) se pueden escribir como.

$$\vec{M}^*\frac{d^2}{dt^2}\vec{y}(t) + \vec{C}^*\frac{d}{dt}\vec{y}(t) + \vec{K}^*\vec{y}(t) = \vec{F}^*(t) \qquad (4.24)$$

que resulta ser un sistema de ecuaciones diferenciales desacoplado, cuya ecuación i-ésima se puede escribir como:

$$m_i^*\frac{d^2}{dt^2}y_i(t) + c_i^*\frac{d}{dt}y_i(t) + k_i^*y_i(t) = f_i^*(t) \qquad (4.25)$$

La Ec 4.25 representa la ecuación de equilibrio dinámico de un sistema de un grado de libertad. Por lo anterior se puede decir que un sistema de N grados de libertad se transforma en N sitemas de un grado de libertad. Los coeficientes de las ecuaciones de un grado de libertad resultan ser:

$$m_i^* = \sum_{k=1}^{N} m_k \left(r_k^i\right)^2 \tag{4.26}$$

$$c_i^* = 2\omega_i \zeta_i \tag{4.27}$$

$$k_i^* = \omega_i^2 m_i^* \tag{4.28}$$

$$f_i^* = -\frac{\sum_{k=1}^{N} m_k r_k^i}{\sum_{k=1}^{N} m_k \left(r_k^i\right)^2} \frac{d^2}{dt^2} u_g(t) = -c_i \frac{d^2}{dt^2} u_g(t) \tag{4.29}$$

en donde:

m_k = masa asociada al grado
. de libertad k-ésimo
r_k^i = componente k-ésimo del
. i-ésimo eigenvector (modo)
ω_i = frecuencia natural de
. vibración del i-ésimo modo
ζ_i = fracción del amortiguamiento
. crítico del i-ésimo modo (4.30)
$c_i = \dfrac{\sum_{k=1}^{N} m_k r_k^i}{\sum_{k=1}^{N} m_k \left(r_k^i\right)^2}$ = coeficiente de
. participación del i-ésimo modo

4.1.3.3 Integración paso a paso de las ecuaciones de movimiento desacopladas

Como las ecuaciones de movimiento desacopladas (Ec 4.25) corresponden a las de un grado de libertad, los métodos de integración son los tradicionales.

- Exacto, para el caso de aproximar la función $f_i^*(t)$ en tramos seccionalmente continuos con una variación lineal (que es lo usual).

- Aproximado, mediante un método numérico como el método de Newmark-Wilson.

El paso de integración se define en el inciso 4.1.2

4.1.3.4 Cuantificación de la respuesta de la estructura

De acuerdo con el inciso anterior para el tiempo de integración considerado se cuantifican, para cada paso de integración, los siguientes vectores.

$$
\begin{aligned}
\vec{y}(t) &= \text{vector de desplazamientos transformado}\\
\frac{d}{dt}\vec{y}(t) &= \text{vector de velocidades transformado}\\
\frac{d^2}{dt^2}\vec{y}(t) &= \text{vector de aceleraciones transformado}
\end{aligned}
\tag{4.31}
$$

Al sustituír las Ec 4.31 en las Ec 4.18 y 4.20 se obtiene la respuesta de la estructura representada por los vectores de desplazamiento relativo, de velocidad relativa, y de aceleración relativa, es decir.

$$
\begin{aligned}
\vec{u}(t) &= \tilde{R}\vec{y}(t)\\
\frac{d}{dt}\vec{u}(t) &= \tilde{R}\frac{d}{dt}\vec{y}(t)\\
\frac{d^2}{dt^2}\vec{u}(t) &= \tilde{R}\frac{d^2}{dt^2}\vec{y}(t)
\end{aligned}
\tag{4.32}
$$

4.1.3.4 Obtención de los elementos mecánicos y cinemáticos de la estructura debidos al sismo

Conocida la historia del vector de desplazamientos de la estructura (según se indica en el inciso anterior) se puede determinar la historia de los elementos mecánicos y cinemáticos en los puntos que se requieran de la estructura.

4.1.4 Método de la respuesta espectral

Este método corresponde al denominado análisis en las NTC para diseño por sismo. Su secuencia se resume a continuación.

4.1.4.1 Solución del problema de valores característicos (eigenvalores) de las ecuaciones de equilibrio dinámico

El procedimiento es el mismo que el descrito en el inciso 4.1.3.1 del método directo de superposición modal.

4.1.4.2 Desacoplamiento de las ecuaciones de equilibrio dinámico

El procedimiento es el mismo que el descrito en el inciso 4.1.3.2 del método directo de superposición modal.

4.1.4.3 Obtención de la respuesta espectral de cada una de las ecuaciones de equilibrio desacopladas

De acuerdo con el RCDF87 se calcula mediante la siguiente expresión.

$$y^{i}_{máx} = c_i \frac{A_i}{\omega_i^2} \tag{4.33}$$

donde:

$$\begin{aligned} y^{i}_{máx} &= \text{respuesta espectral de desplazamientos transformados del modo i-ésimo} \\ \omega_i &= \text{Frecuencia natural de vibración del modo i-ésimo} \\ A_i &= \text{Ordenada del espectro de aceleraciones de diseño asociada al período natural de vibración } T_i = \frac{2\pi}{\omega_i} \\ c_i &= \text{Coeficiente de participación del modo i-ésimo} \end{aligned} \tag{4.34}$$

4.1.4.4 Cuantificación de los vectores de respuesta máximos de la estructura para cada modo

De acuerdo con la Ec 4.32a, el vector de desplazamientos máximo de la estructura, correspondiente al modo i-ésimo, resulta ser.

$$\vec{u}_{i\,máx} = \vec{r}^{\,i} y^{i}_{máx} \tag{4.35}$$

donde:

$$\vec{r}^{\,i} = \text{Eigenvector asociado al modo i-ésimo} \qquad (4.36)$$

De acuerdo con la Ec 4.36, a cada modo de la estructura le corresponde un vector de desplazamientos máximo. Con base en la formulación de las ecuaciones de equilibrio de las estructuras, a cada vector de desplazamientos le corresponden un conjunto de elementos mecánicos y cinemáticos (fuerzas normales, fuerzas cortantes, momentos flexionantes, momentos de volteo, desplazamientos relativos, etc.)

4.1.4.5 Obtención de la respuesta total de la estructura

Una vez conocidos los elementos mecánicos y cinemáticos (fuerzas normales, fuerzas cortantes, momentos flexionantes, momentos de volteo, desplazamientos relativos, etc.) asociadas a cada modo, representado por S_i, para obtener la respuesta de la estructura, representada por S, se procede como se indica a continuación.

4.1.4.5.1 Método de la raíz cuadrada de la suma de los cuadrados (SRSS)

$$S = \sqrt{\sum_{i=1}^{N} S_i^2} \qquad (4.37)$$

4.1.4.5.2 Método de la combinación cuadrática completa (CQC)

$$S = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} S_i p_{ij} S_j} \qquad (4.38)$$

donde:

$$p_{ij} = \frac{8\sqrt{\zeta_i\zeta_j\omega_i\omega_j}\,(\zeta_i\omega_i + \zeta_j\omega_j)\,\omega_i\omega_j}{(\omega_i^2 - \omega_j^2)^2 + 4\zeta_i\zeta_j\omega_i\omega_j(\omega_i^2 + \omega_j^2) + 4(\zeta_i^2 + \zeta_j^2)\,\omega_i^2\omega_j^2} \qquad (4.39)$$

i Valor del amortiguamiento crítico del modo i-ésimo (que se supone consante para todos los modos)

i frecuencia natural de vibración del modo i-ésimo

4.2 Análisis estático

Las NTC para Diseño por Sismo del RCDF87 proponen un método relativamente simple para cuantificar las fuerzas horizontales que un sismo de diseño ocasiona a una edificación cuya altura no exceda de 60 m.

4.2.1 Distribución de las aceleraciones horizontales

De acuerdo con el inciso 8.1 de las NTC para Diseño por Sismo del RCDF87, la hipótesis sobre la distribución de aceleraciones en las masas de las edificaciones se muestra en la Fig 4.2. Para la masa del nivel i-ésimo, la fuerza que la distribución de aceleraciones le ocasiona a la masa se puede escribir como.

$$F_i = m_i \ddot{u}_i = \frac{W_i}{g} \ddot{u}_i \tag{4.40}$$

donde se definen los componentes respectivos.

$$\begin{aligned} F_i &= \text{Fuerza horizontal del nivel i-ésimo} \\ m_i &= \text{masa del nivel i-ésimo} \\ W_i &= m_i g = \text{peso del nivel i-ésimo} \\ \ddot{u}_i &= \text{aceleración del nivel i-ésimo} \end{aligned} \tag{4.41}$$

De acuerdo con la Fig 4.2, la expresión de la aceleración de la masa i-ésima resulta ser.

$$\ddot{u}_i = \frac{h_i}{h_n} \ddot{u}_n \tag{4.42}$$

Al sustituír la Ec 4.42 en la Ec 4.40 se obtiene.

$$F_i = \frac{\ddot{u}_n}{g h_n} W_i h_i \tag{4.43}$$

4.2.2 Fuerzas sísmicas horizontales

De acuerdo con la definición de fuerza cortante basal, se puede expresar la siguiente ecuación.

$$V_0 = \sum_{i=1}^{N} F_i = \frac{\ddot{u}_n}{g h_n}\left(\sum_{i=1}^{N} W_i h_i\right) \tag{4.44}$$

Al considerar la definición de coeficiente sísmico, c, se puede escribir la siguiente expresión.

$$c = \frac{V_0}{W_0} = \frac{\frac{\ddot{u}_n}{g h_n}\left(\sum_{i=1}^{N} W_i h_i\right)}{\sum_{i=1}^{N} W_i} \tag{4.45}$$

Con base en la Ec 4.45 se obtiene la expresión siguiente.

$$\frac{\ddot{u}_n}{g h_n} = c\frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i} \tag{4.46}$$

Al sustituír la Ec 4.46 en la Ec 4.43, la expresión de la fuerza sísmica estática se puede expresar como.

$$F_i = c\frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i} W_i h_i \tag{4.47}$$

4.2.3 Estimación del período fundamental de la estructura

Las NTC para diseño por sismo del RCDF87 recomienda una expresión para estimar el período de vibración del primer modo, T_1, de acuerdo con la modelación estructural a base de rigideces de entrepiso , según se indica a continuación.

a) Los datos de partida se muestran en la Fig 4.2 y son.

$$k_i = \text{Rigideces del entrepiso i-ésimo} \qquad W_i = \text{Pesos del nivel i-ésimo} \tag{4.48}$$

b) Cuantificación de las fuerzas sísmicas, F_i, de cada nivel de acuerdo con la Ec 4.47.

c) Cuantificación de ls fuerzas cortantes, V_i, de cada entrepiso.

$$V_i = \sum_{k=i}^{N} F_k \tag{4.49}$$

d) Obtención de los desplazamientos, u_i, asociados a las fuerzas cortantes de entrepiso.

$$\Delta u_i = \frac{V_i}{k_i} \tag{4.50}$$

e) Otención de los desplazamientos, x_i, que provocan las fuerzas sísmicas, con base en la Ec 4.50.

$$x_1 = 0 \qquad x_i = x_{i-1} + \Delta u_i \qquad \forall\, i = 2 \ldots N \tag{4.51}$$

f) Obtención de las aceleraciones armónicas correspondientes a los desplazamientos del inciso anterior (inciso e), asociados a la frecuencia natural de vibración, $_1$.

$$\ddot{x}_i = \omega_1^2 x_i \tag{4.52}$$

g) obtención de las fuerzas dinámicas asociadas a las aceleraciones armónicas del inciso anterior (inciso f).

$$F_{ar} = m_i \ddot{x}_i = \frac{W_i x_i}{g} \omega_1^2 \tag{4.53}$$

h) Cuantificación de los trabajos que realizan las fuerzas F_i(Ec 4.47) y F_{ar} (Ec. 4.53) debido a los desplazamientos x_i (Ec 4.51).

$$W_{Fi} = \sum_{i=1}^{N} F_i x_i$$

$$W_{Far} = \frac{\omega_1^2}{g} \sum_{i=1}^{N} W_i x_i^2 \tag{4.54}$$

i) Obtención de la frecuencia natural de vibración T_1, al igualar los trabajos dados por las Ec 4.54.

$$\omega_1^2 = g \frac{\sum_{i=1}^{N} F_i x_i}{\sum_{i=1}^{N} W_i x_i^2}$$

$$T_1 = \frac{2\pi}{\omega_1} = 2\pi \sqrt{\frac{\sum_{i=1}^{N} W_i x_i^2}{g \sum_{i=1}^{N} F_i x_i}} \tag{4.55}$$

4.2.4 Reducción de las fuerzas cortantes estáticas

Las NTC para diseño por sísmo del RCDF87 establecen que las fuerzas sismicas descritas en el inciso 4.2.2, obtenidas con la Ec 4.55, pueden adoptarse valores los menores que se indica a continuación.

a) El período fundamental de vibración se obtiene con la Ec 4.55.

b) Si $T_1 \leq T_b$ el valor del coeficiente sismico, c, en la Ec 4.47 se sustituye por el valor de la ordenada del espectro de aceleraciones, a, dado por la Ec 3.38, y resulta ser.

$$F_i = a \frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i} W_i h_i \tag{4.56}$$

c) Si $T_1 > T_b$ las fuerzas sísmicas se cuantifican con las expresiones siguientes.

$$F_i = aW_i(k_1 h_i + k_2 h_i^2) \qquad \forall\, a \geq \frac{c}{4} \tag{4.57}$$

donde:

$$k_1 = q[1 - r(1 - q)]\frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i}$$

$$k_2 = 1.5rq(1 - q)\frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i^2} \tag{4.58}$$

$$q = \left(\frac{T_b}{T_1}\right)^r$$

4.3 Método simplificado

Las NTC para diseño por sismo establece el cumplimiento simultáneo de las siguientes condiciones para que sea aplicable el denominado método simplificado de análisis.

4.3.1 Consideraciones generales

I. En cada planta, al menos el 75 por ciento de las cargas veticales están soportadas por muros ligados entre sí mediante losas monolíticas u otros sistemas de piso suficientemente resistentes y rígidos al corte. Dichos muros tendrán distribución sensiblemente simétrica con respecto a dos ejes ortogonales y deben satisfacer las condiciones que establecen las NTC correspondientes. Es admisible cierta asimetría en la distribución de los muros cuando existan en todos los pisos dos muros de cargas perimetrales paralelos, cada uno con longitud al menos igual a la mitad de la dimensión mayor en planta del edificio. Los muros a que se refiere este párrafo pueden ser de mamposteria, concreto reforzado o madera; en este último caso deben estar arriostrados con diagonales.

II. La relación entre longitud y ancho de la planta del edificio no excede de 2.0 a menos que, para fines de análisis sísmico, se pueda suponer dividida dicha planta en tramos independientes cuya relación longitud a anchura satisfaga esta restricción y cada tramo resista según el criterio que se indica en la tabla 7.1 de las NTC para diseño por sismo.

III. La relación entre la altura y la relación mínima de la base del edificio no excede a 1.5 y la altura del edificio no es mayor de 13 m.

4.3.2 Consideraciones específicas

Para aplicar este método se hace caso omiso de los desplazamientos horizontales, torsiones y momentos de volteo.

Se debe verificar únicamente que en cada piso la suma de las resistencias al corte de los muros de carga, proyectados en la dirección en que se considera la aceleración , sea cuando menos igual a la fuerza cortante total que obre en dicho piso, calculada según se especifica en el inciso 4.2.2.

Los coeficientes sísmicos que se deben emplear se indican en la tabla 7.1 de las NTC para diseño por sismo, correspondientes a las construcciones del grupo B. Para las construcciones del grupo A dichos coeficientes se deven multiplicar por 1.5.

Tabla 7.1 Coeficientes sísmicos reducidos para el método simplificado, correspondiente a estructuras del grupo B (NTC para diseño por sismo RCDF87).

ZONA	MUROS DE PIEZAS MACISAS O DIAFRAGMAS DE MADERA CONTRACHAPEADA			MUROS DE PIEZAS HUECAS O DIAFRAGMAS DE DUELAS DE MADERA*		
	ALTURA DE LA CONSTRUCCION (m)			ALTURA DE LA CONSTRUCCION (M)		
	$H<4$	$4 \le H \le 7$	$7<H \le 13$	$4<H$	$4 \le H \le 7$	$7<H \le 13$
I	0.07	0.08	0.08	0.10	0.11	0.11
IIyIII	0.13	0.16	0.19	0.15	0.19	0.23

* Diafragmas de duelas de madera inclinadas o sistemas de muros formados por duelas de madera verticales u horizontales arriostradas con elementos de madera maciza.

4.3.3 Consideraciones de las NTC para diseño y construcción de estructuras de mampostería

En el inciso 4.1.3 de las NTC para diseño y construcción de estructuras de mamposteria se establece lo siguiente.

El análisis para la determinación de los efectos de las cargas laterales debidas a sismo se hace con base en las rigideces relativas de los distintos muros. Estas de determinan tomando en cuenta las deformaciones de cortante y de flexión. Para estas últimas se considera la sección transversal agrietada del muro cuando la relación de carga vertical a momento flexionante es tal que se presentan tensiones verticales. Se debe tomar en cuenta la restricción que impone a la rotación de los muros la rigidez de los sistemas de piso y techo y la de los dinteles.

Es admisible considerar que la fuerza cortante que toma cada muro es proporcional a su área transveral, ignorar los efectos de torsión y de momento de volteo.

La contribución a la resistencia a fuerzas cortantes de los muros cuya relación de altura de entrepiso, H, a longitud, L, es mayor que 1.33 se debe reducir al multiplicar la resistencia por el coeficiente $(1.33\ L/H)^2$.

4.4 Reducción de fuerzas sísmicas

Las NTC para diseño por sísmo del RCDF87 establecen que las fuerzas sismicas descritas en los incisos 4.1 y 4.2 se pueden reducir al dividirlas entre el factor reductivo Q'.

4.4.1 Estructuras regulares

Para las estructuras que satisfacen las condiciones de regularidad indicadas en el inciso 4.2.4, Q' se obtiene con las siguientes expresiones.

$$
\begin{aligned}
Q' &= Q \qquad \text{si T se desconoce} \\
Q' &= Q \qquad\qquad \forall\ T \geq T_a \\
Q' &= 1 + \frac{T}{T_a}(Q-1) \qquad \forall\ T < T_a
\end{aligned}
\tag{4.59}
$$

donde:

a) T es igual al período fundamental de vibración (inciso 4.2.3) cuando se emplee el método estático (inciso 4.2.2) e igual al período de natural de vibración del modo que se considere cuando se emplee el método de análisis modal (inciso 4.1.4).

b) T_a es un período caracerístico del espectro de diseño utilizado (inciso 4.2.6).

c) Los desplazamientos de diseño sísmico se obtienen al multiplicar por el factor de comportamiento sísmico, Q, a los desplazamientos obtenidos con las fuerzas sísmicas reducidas.

d) Cuando se adopten dispositivos especiales capaces de disipar energía por amortiguamiento o comportamiento inelástico, se pueden emplear criterios de diseño sísmico que difieran de los aquí especificados, pero congruentes con ellos, con la aceptación del DDF.

4.4.2 Estructuras irregulares

Para las estructuras que no satisfacen las condiciones de regularidad indicadas en el inciso 4.2.4, Q' se obtiene con las expresiones del inciso anterior (Ec 4.59) multiplicado por 0.8.

4.5 Efectos de torsión

Las NTC para diseño por sísmo del RCDF87 establecen que para fines de diseño, el momento torsionante se debe tomar por lo menos igual a la fuerza cortante de entrepiso multiplicada por la excentricidad que para cada marco o muro resulte mas desfavorable de ls siguientes

$$\begin{aligned} e_d &= 1.5e_s + 0.1b \\ e_d &= e_s - 0.1b \end{aligned} \tag{4.60}$$

donde:

e_s = Excentricidad torsional de rigideces calculada del entrepiso, igual a la distancia entre el centro de torsión del nivel correspondiente y la fuerza cortante en dicho nivel.

b = Dimensión de la planta que se considera, medida en la dirección de e_s.

La excentridicidad de diseño, e_d, en cada sentido no se debe tomar menor que la mitad del máximo valor de la excentricidad calculada, e_s, para los entrepisos que se hallen abajo del que se considera, ni se debe tomar el momento torsionante de ese entrepiso menor que la mitad del máximo calculado para los entrepisos que están arriba del considerado.

4.6 Efectos de segundo orden

Las NTC para diseño por sísmo del RCDF87 establecen que se deben tomar en cuenta explícitamente en el análisis los efectos de segundo orden, esto es, los momentos y cortantes adicionales provocados por las cargas verticales al obrar en la estructura desplazada lateralmente, en toda estructura en que la diferencia en desplazamientos laterales entre dos niveles consecutivos, u_i, dividida entre la diferencia de altura correspondientes, h_i, es tal que:

$$\frac{\Delta u_i}{h_i} > 0.08 \frac{V}{W} \qquad (4.62)$$

donde

V = Fuerza cortante en el entrepiso considerado

W = Peso de la construcción encima del entrepiso

El peso de la construccion incluye cargas muertas y vivas.

4.7 Efectos bidireccionales

Las NTC para diseño por sísmo del RCDF87 establecen que los efectos de ambos componentes horizontales del movimiento del terreno se deben combinar al tomar en cada dirección en que se analice la estructura, el 100 % de los efectos del componente que obra en esa dirección y el 30 % de los efectos del que obra perpendicularmente a ella, con los signos que para cada concepto resulten mas desfavorables.

5. FUERZAS SISMICAS EN LOS ELEMENTOS ESTRUCTURALES RESISTENTES DE LAS EDIFICACIONES

El concepto de fuerzas sísmicas en elementos estructurales resistentes de una edificación es la manera de especificar la magnitud de las fuerzas sísmicas que actúan en cada uno de los elementos estructurales resistentes en los métodos que utilizan simplificaciones estructurales para cuantificar las fuerzas sísmicas.

5.1 En los modelos estructurales donde se utilizan las ecuaciones de equilibrio dinámico de las edificaciones

En los modelos estructurales que formulan las ecuaciones de equilibrio a través del concepto de subestructuras unidas a un diafragma (nivel), rígido o no, la información que se maneja de manera sistemática es el equilibrio de cada uno de los elementos estructurales que la forman. Entonces, el concepto de fuerzas sismicas en los elementos estructurales es transparente ya que se cuenta con la información integral de cada uno de los elementos estructurales de la edificación, al establecer las ecuaciones de equilibrio.

5.2 En los modelos estructurales donde se utiliza el concepto de rigidez de entrepiso

El modelo donde se emplea el concepto de rigidez de entrepiso es el modelo mas simple donde se utiliza el concepto de diafragma rigido. Es un modelo en extinción ya que los modelos a que hace referencia el inciso 5.1 son mas generales. Se presenta porque el RCDF87 hace referencia a algunos conceptos que utiliza. Se basa en las siguientes hipótesis:

a) Se considera el equilibrio en un solo diafragma (nivel) rígido en donde la carga que actúa es la fuerza cortante en el entrepiso correspondiente, localizada en su centro de masas.

b) Las fuerzas que resisten a la fuerza cortante las proporcionan las rigideces de entrepiso (resortes) del entrepiso correspondiente que definen el centro de torsión (o de rigideces).

c) Las rigideces de entrepiso las forman los marcos (o muromarcos) planos, sensiblemente paralelos en dos direcciones ortogonales.

d) En los desarrollos que siguen se considera que el edificio tiene una distribución de rigideces regular en elevación. Es decir, que las columnas de un diafragma (nivel) únicamente están unidas con niveles consecutivos.

En la Fig 5.1 se muestra la idealización del modelo estructural descrito en los incisos anteriores.

5.2.1 Centro de rigideces (de torsión) del entrepiso

Debido a que los elementos resistentes de un entrepiso se representan mediante las rigideces del mismo, se define como centro de rigidez (o de torsión) al punto en donde al actuar las fuerzas cortantes únicamente provocan desplazamientos lineales.

5.2.1.1 Fuerzas cortantes directas en los resortes paralelos al eje y de referencia

Con base en la Fig 5.2, la fuerza que soporta cada resorte (rigidez de entrepiso) paralelo al eje y resulta ser

$$V_{jy}^{d} = k_{jy} v \tag{5.1}$$

De acuerdo con la condición de equilibrio de fuerzas paralelas al eje y se puede escribir como.

$$V_y = \sum_{j=1}^{NX} V_{jy}^{d} = v \sum_{j=1}^{NX} k_{jy} \tag{5.2}$$

Con base en las Ec 5.1 y 5.2 se obtienen las siguientes expresiones.

$$v = \frac{V_y}{\sum_{j=1}^{NX} k_{jy}} \tag{5.3}$$

$$V_{jy}^d = \frac{k_{jy}}{\sum_{j=1}^{NX} k_{jy}} V_y \tag{5.4}$$

5.2.1.2 Fuerzas cortantes directas en los resortes paralelos al eje x de referencia

Al seguir un razonamiento similar al inciso 5.2.1.1 y utilizar la Fig 5.3 se obtienen las siguientes ecuaciones.

$$V_{ix}^d = k_{ix} u \tag{5.5}$$

$$V_x = \sum_{i=1}^{NY} V_{ix}^d = u \sum_{i=1}^{NY} k_{ix} \tag{5.6}$$

$$u = \frac{V_x}{\sum_{i=1}^{NY} k_{ix}} \tag{5.7}$$

$$V_{ix}^d = \frac{k_{ix}}{\sum_{i=1}^{NY} k_{ix}} V_x \tag{5.8}$$

5.2.1.3 Coordenadas del centro de torsión

Se denomina centro de torsión (CT) o centro de rigideces (CR) al punto localizado sobre el diafragma rígido donde al actuar la fuerza cortante correspondiente únicamente le provoca desplazamientos lineales.

Al aplicar la definición de CT a la fuerza cortante paralela al eje y, al establecer el equilibrio de momentos resulta.

$$\begin{aligned} x_t V_y &= \sum_{j=1}^{NX} x_j v_{jy}^d \\ &= \sum_{j=1}^{NX} x_j \frac{k_{jy}}{\sum_{j=1}^{NX} k_{jy}} V_y \\ &= \frac{\sum_{j=1}^{NX} x_j k_{jy}}{\sum_{j=1}^{NX} k_{jy}} V_y \end{aligned} \tag{5.9}$$

De acuerdo con la Ec 5.9 se obtiene la expresión de la abscisa del centro de torsión.

$$x_t = \frac{\sum_{j=1}^{NX} x_j k_{jy}}{\sum_{j=1}^{NX} k_{jy}} \tag{5.10}$$

Al aplicar la definición de CT a la fuerza cortante paralela al eje x, se obtiene la siguiente expresión de la ordenada del centro de torsión.

$$y_t = \frac{\sum_{i=1}^{NY} y_i k_{ix}}{\sum_{i=1}^{NY} k_{ix}} \tag{5.11}$$

5.2.2 Excentricidades

Las fuerzas sísmicas asociadas a los diafragmas rígidos actúan en el punto denominado centro de masas (CM) y no en el centro de torsión, que pueden ser diferentes. A las distancias paralelas a la dirección de las fuerzas cortantes se les denominan excentricidades.

5.2.2.1 Excentricidades calculadas

Las excentricidades correspondientes a las dos fuerzas cortantes ortogonales se pueden escribir como.

$$e_{sx} = |x_m - x_t|$$

donde :

e_{sx} = Excentricidad de la fuerza Cortante V_y (5.12)

x_m = Abscisa del centro de masas

x_t = Abscisa del centro de torsión

$$e_{sy} = |y_m - y_t|$$

donde :

e_{sy} = Excentricidad de la fuerza Cortante V_x (5.13)

y_m = Ordenada del centro de masas

y_t = Ordenada del centro de torsión

5.2.2.2 Excentricidades de diseño

Las NTC para diseño por sismo del RCDF87 establecen que a cada excentricidad calculada se le debe asociar dos excentricidades de diseño, según se indica a continuación.

a) Excentricidades asociadas a la fuerza cortante V_y.

$$\begin{aligned} e_{dx} &= 1.5e_{sx} + 0.1b_x \\ e_{dx} &= e_{sx} - 0.1b_x \end{aligned} \qquad (5.14)$$

donde:

b_x es la dimensión de la planta que se considera medida en la dirección de e_{sx} (perpendicular a la fuerza cortante V_y).

b) Excentricidades asociadas a la fuerza cortante V_x.

$$e_{dy} = 1.5e_{sy} + 0.1b_y$$
$$e_{dy} = e_{sy} - 0.1b_y \tag{5.15}$$

donde:

b_y es la dimensión de la planta que se considera medida en la dirección de e_{sy} (perpendicular a la fuerza cortante V_x).

5.2.3 Fuerzas cortantes debidas a la torsión

De acuerdo con el inciso 5.2.2 para efectos de diseño se deben considerar los efectos de un momento torsionante, M, cuantificado con las siguientes expesiones.

$$M = M_{ty} = e_{dx}V_y$$
$$= M_{tx} = e_{dy}V_x \tag{5.16}$$

Con base en la Fig 5.4 se puede afirmar que el momento torsionante se equilibra con las fuerzas cortantes que provoca en todos los resortes. El movimiento de cuerpo rígido que el par torsionante le provoca al diafragma rígido es el giro, .

Los desplazamientos lineales en los resortes paralelos a cada uno de los ejes de referencia, al considerar que el desplazamiento angular es pequeño, de tal manera que el seno y la tangente del mismo se pueda aproximar por el valor del ángulo, resultan ser.

$$u_i = \theta\overline{y}_i$$
$$v_j = \theta\overline{x}_j \tag{5.17}$$

donde:

$$\overline{x}_i = x_i - x_t$$
$$\overline{y}_j = y_j - y_t \tag{5.18}$$

Las fuerzas cortantes debidas al par torsionante resultan ser.

$$
\begin{aligned}
V_{ix}^{t} &= k_{ix}u_i = \theta k_{ix}\overline{y}_i \\
V_{jy}^{t} &= k_{jy}v_j = \theta k_{jy}\overline{x}_j
\end{aligned}
\tag{5.19}
$$

Al establecer el equilibrio de pares respecto al centro de torsión se obtiene que.

$$
\begin{aligned}
M &= \sum_{i=1}^{NY} V_{ix}^{t}\overline{y}_i + \sum_{j=1}^{NX} V_{jy}^{t}\overline{x}_j \\
&= \theta\left[\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2\right]
\end{aligned}
\tag{5.20}
$$

De las Ec 5.20 se obtiene el valor del desplazamiento angular de cuerpo rígido.

$$
\theta = \frac{M}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2}
\tag{5.21}
$$

Al sustituír la Ec 5.21 en las Ec 5.19 se obtienen las expresiones de las fuerzas cortantes que el momento torsionante ocasiona a los resortes (rigideces de entrepiso).

$$
V_{ix}^{t} = \frac{k_{ix}\overline{y}_i}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M
\tag{5.22}
$$

$$
V_{jy}^{t} = \frac{k_{jy}\overline{x}_j}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M
\tag{5.23}
$$

5.2.4 Fuerzas cortantes de diseño en los resortes (rigideces de entrepiso)

Con base en los desarrollos de los incisos anteriores, la fuerza cortante que cada resorte (rigidez de entrepiso) soporta es la suma de la fuerza cortante directa mas la fuerza cortante debida a la torsión, como se expresa a continuación.

$$V_{ix} = V_{ix}^{d} + V_{ix}^{t}$$
$$V_{jy} = V_{jy}^{d} + V_{jy}^{t} \qquad (5.24)$$

Para cuantificar la Ec 5.24a se hace uso de ls Ec 5.8 y 5.22, mientras que para la Ec 5.24b se utilizan las Ec 5.4 y 5.23.

5.3 En el método simplificado

En este método se hace caso omiso del efecto de torsión, por lo que únicamente se consideran las fuerzas cortantes directas.

Las NTC para diseño y construcción de estructuras de mampostería establece que es admisible considerar que la fuerza cortante que toma cada muro es proporcional a su área transversal

6. EJEMPLOS DESARROLLADOS PASO A PASO

En este capítulo se presentan los ejemplos que permiten aplicar los conceptos descritos en este curso. Los ejemplos, por tratar de aplicar paso a paso los aspectos operativos de los métodos, corresponden únicamente a métodos que se pueden desarrrollar sin un número exagerado de operaciones, de tal manera que se pueden llevar a cabo con calculadora, lápiz y papel.

6.1 Edificación utilizada

En la Fig 6.1 se muestra la planta y elevación de un edificio de interés social que sirve de base para llevar a cabo los ejemplos de aplicación. Las particularidades del edificio se indican a continuación.

6.1.1 Uso de las edificaciones

Con base en el inciso 3.1 , el uso de la edificación es vivienda, por lo que le corresponde el Grupo B.

Por tratarse de una edificación de 667 $m^2 < 6000\ m^2$, con una altura de 12.5 m < 30 m, se ubica en el subgrupo B2.

6.1.2 Zonificación sísmica

La edificación se localiza en la zona I.

6.1.3 Coeficiente sísmico

De acuerdo con el inciso 3.2, y los datos especificados en los incisos 6.1.1 y 6.1.2, el coeficiente sismico que le corresponde a la edificación es c = 0.16.

6.1.4 Condiciones de regularidad

Con base en los datos de la edificación (Fig 6.1) se obtienen los siguientes parámetros en relación con el inciso 3.4, a fin de definir el coeficiente de reducción de las fuerzas sísmicas, Q'.

a) Planta sensiblemente simétrica respecto a dos ejes ortogonales (respecto a masas y elementos resistentes).

b) Altura/dimensión menor en planta = 12.5/8.4 = 1.49 < 2.5.

c) Largo/ancho = 15.9/8.4 = 1.9 < 2.5.

d) De acuerdo con la tabla 6.1 la relación entre los pesos de los niveles superior a inferior es igual a uno, con excepción del quinto nivel (último) que es igual a 0.88.

e) Todos los pisos tienen la misma área, igual a 133.56 m^2.

f) En relación con los conceptos de rigidez al corte y excentricidades se discuten en los incisos correspondientes.

6.1.5 Factor de comportamiento sísmico

La resistencia a las fuerzas laterales se suministra por:

a) muros de mampostería de piezas huecas.
b) confinadas en toda la altura.
c) de 15 cm de espesor.
d) resistencia al esfuerzo cortante de 2.5 kg/cm^2.

Con base en el inciso 3.5, el factor de comportamiento asociado a las dos direcciones ortogonales resultan ser.

$Q_x = 1.5$

$Q_y = 1.5$

6.1.6 Espectro de diseño

Con base en el inciso 3.6 y la tabla 3.1 de la NTC para diseño por sismo, los parámetros del espectro de respuesta de diseño en la zona I junto con el coeficiente sísmico especificado en el inciso 6.1.3, resultan ser.

T_a = 0.2 s

T_b = 0.6 s

r = 1/2

6.2 Análisis estático

De acuerdo con el inciso 4.2.2 las fuerzas horizontales que un sísmo de diseño ocasiona a una edificación están dadas por la Ec 4.39, reproducida a continuación.

$$F_i = c\frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i} W_i h_i \qquad (4.39)$$

En este método no es necesario hacer uso de un modelo estructural para el edificio, excepto si se desea estimar el período fundamental del mismo.

6.2.1 Fuerzas cortantes

Con base en los datos de la geometría y pesos del edificio, así como los datos especificados en el inciso 6.1, los elementos de la Ec 4.39 se resumen en la tabla 6.1.

Tabla 6.1 Fuerzas sísmicas (método estático)

Nivel	W_i (t)	h_i (m)	$W_i h_i$ (tm)	F_i (t)	V_i (t)
5	91.2	12.5	1140.0	24.73	24.73
4	104.0	10.9	1040.0	22.57	47.30
3	104.0	7.5	780.0	16.92	64.22
2	104.0	5.0	520.0	11.28	75.50
1	104.0	2.5	260.0	5.64	81.14
Σ	507.2		3740.0		

De acuerdo con los valores de las columnas 2 y 4 de la tabla 6.1 se puede cuantificar el siguiente coeficiente.

$$C\frac{\sum_{i=1}^{N} W_i}{\sum_{i=1}^{N} W_i h_i} = 0.16*\frac{507.2}{3740.0} = 0.0217 \qquad (6.1)$$

6.2.2 Estimación del periodo fundamental de vibración

De acuerdo con el inciso 4.2.3 la estimación del período fundamental se obtiene mediante la Ec 4.47b, reproducida a continuación.

$$T_1 = 2\pi\sqrt{\frac{\sum_{i=1}^{N} W_i x_i^2}{g\sum_{i=1}^{N} F_i x_i}} \qquad (4.47b)$$

Los valores especificados en las tablas 6.1, 6.5 y 6.6 sirven de base para la cuantificación de la Ec 4.47b.

6.2.2.1 En la dirección del eje x

Las operaciones numéricas para determinar los elementos de la Ec 4.47b se resumen en la tabla 6.2.

Tabla 6.2 Estimación del período fundamental, T_{1x}, en la dirección del eje x

Nivel	k_{ix} (t/cm)	u_{ix} (m)	x_i (m)	$F_i x_i$ (tm)	$W_i x_i^2$ (tm²)
5	203.65	0.00121	0.00531	0.13132	0.00257
4	372.46	0.00127	0.00410	0.09254	0.00175
3	528.42	0.00122	0.00283	0.04788	0.00083
2	749.62	0.00101	0.00161	0.01816	0.00027
1	1363.69	0.00060	0.00060	0.00338	0.00004
Σ				0.29328	0.00546

Al sustituír los valores de las columnas 5 y 6 de la tabla 6.2 en la Ec 4.47b resulta.

$$T_{1x} = 6.28\sqrt{\frac{0.00546}{9.81*0.29328}} = 0.2736\ s \qquad (6.2)$$

6.2.2.2 En la direccion del eje y

Las operaciones numéricas para determinar los elementos de la Ec 4.47b se resumen en la tabla 6.3.

Tabla 6.3 Estimación del período fundamental, T_{1y}, en la dirección del eje y

Nivel	k_{iy} (t/cm)	u_{iy} (m)	x_i (m)	$F_i x_i$ (tm)	$W_i x_i^2$ (tm²)
5	65.93	0.0037	0.0159	0.3932	0.0231
4	121.28	0.0039	0.0122	0.2754	0.0155
3	173.85	0.0037	0.0083	0.1404	0.0072
2	253.15	0.0030	0.0046	0.0519	0.0022
1	515.28	0.0016	0.0016	0.0091	0.0003
Σ				0.8699	0.0483

Al sustituír los valores de las columnas 5 y 6 de la tabla 6.3 en la Ec 4.47b resulta.

$$T_{1y} = 6.28\sqrt{\frac{0.0483}{9.81*0.8699}} = 0.4724\ s \qquad (6.3)$$

6.2.3 Factores reductivos de las fuerzas sísmicas

De acuerdo con el inciso 4.3 los factores reductivos de las fuerzas sismicas resultan ser.

6.2.3.1 Factor reductivo para fuerzas paralelas al eje x

Al comparar el período fundamental T_{1x} con el valor de T_a resulta.

$$T_{1x} = 0.2736 > T_a = 0.2$$

$$\text{donde:} \qquad (6.4)$$

$$Q_x' = Q_x = 1.5$$

6.2.3.2 Factor reductivo para fuerzas paralelas al eje y

Al comparar el período fundamental T_{1y} con el valor de T_a resulta.

$$T_{1y} = 0.4724 > T_a = 0.2$$

$$\text{donde:} \qquad (6.5)$$

$$Q_y' = Q_y = 1.5$$

6.2.4 Fuerzas sismicas reducidas

Al dividir las fuerzas sísmicas estáticas de la tabla 6.1 entre los correspondientes factores reductivos dados por las Ec 6.4 y 6.5 se obtienen las fuerzas sísmicas reducidas de la tabla 6.4.

Tabla 6.4 Fuerzas sísmicas estáticas sin reducir y reducidas

Nivel	**F_i (t)**	**V_i (t)**	**F_{1xr} (t)**	**V_{1xr} (t)**	**F_{1yr} (t)**	**V_{1yr} (t)**
5	24.73	24.73	16.48	16.48	16.48	16.48
4	22.57	47.30	15.05	31.53	15.05	31.53
3	16.92	64.22	11.28	42.81	11.28	42.81
2	11.28	75.50	7.52	50.33	7.52	50.33
1	5.64	81.14	3.76	54.09	3.76	54.09

6.2.5 Reducción de las fuerzas cortantes con base en el período fundamental de vibración

De acuerdo con el inciso 4.2.4 existe la posibilidad de reducir las fuerzas sísmicas de la tabla 6.4, con base en el valor de los períodos fundamentales de vibración.

6.2.5.1 En la direccion del eje x

Al ubicar el período fundamental en el espectro de diseño sismico se tiene que.

$$T_a = 0.2 < T_{1x} = 0.2736 < T_b = 0.6 \tag{6.6}$$

De acuerdo con la Ec 6.6 se concluye que no deben reducirse las fuerzas estáticas en la dirección del eje x de la tabla 6.4.

6.2.5.2 En la direccion del eje y

Al ubicar el período fundamental en el espectro de diseño sismico se tiene que.

$$T_a = 0.2 < T_{1y} = 0.4724 < T_b = 0.6 \tag{6.7}$$

De acuerdo con la Ec 6.7 se concluye que no deben reducirse las fuerzas estáticas en la dirección del eje y de la tabla 6.4.

6.3 Método dinámico (análisis modal espectral)

Este método se describe en el inciso 4.1.4 y su aplicación implica un modelo estructural para el edificio.

6.3.1 Modelo estructural del edificio

En este ejemplo se utiliza el modelo estructural descrito en el inciso 2.4.5, construído a base de subestructuras formadas con rigideces de entrepiso (resortes) unidas con diafragmas rígidos. Este modelo no es el recomendable, pero se utiliza porque permite ejemplificar algunos conceptos del RCDF87 y el número de operaciones que se tienen que realizar resultan ser mucho menor que el de los modelos donde se utiliza una computdora.

El modelo estructural del edificio se construye mediante subestructuras planas formados por muros planos, construídos con mampostería. La definición de los muros planos se hace en las dos direcciones ortogonales en que están orientados los ejes de la planta del edificio. Los 9 ejes letra (muros 1-x, 2-x, 3-x, 4-x, 5-x, 6-x, 7-x, 8-x y 9-x) y los 3 ejes número (muros 1-y, 2-y y 3-y).

En las Fig 6.2 y 6.3 se muestran las idealizaciones de los muros planos mediante rigideces de entrepiso, y en la Fig 6.4 se representan los dos modelos estructurales del edificio asociados a las dos direcciones ortogonales. Cada estructura unidimensional tiene 5 grados de libertad.

Las rigideces de entrepiso de los muros planos se determinaron con el método del elemento finito, al considerar que actúa un sistema de fuerzas horizontales igual al que proporciona el método estático (inciso 6.2). Los valores que resultan se muestran en las Fig 6.2 y 6.3, así como en las tablas 6.5 y 6.6.

6.3.2 Solución del problema de valores característicos

Las formas modales (eigenvectores) y las correspondientes frecuencias naturales de vibración (eigenvalores), según el inciso 4.1.3.1, se pueden obtener con métodos que utilicen calculadoras o computadoras. En este ejemplo el problema de valores característicos se resolvió al utilizar el método matricial de Jacobi. Las matrices de rigideces y de masas para cada modelo unidimensional se construyen como se indica en las Ec 6.8 y 6.9.

Tabla 6.5 Rigideces de entrepiso de los muros paralelos al eje x (t/cm)

Entrepis	**1-x**	**2-x**	**3-x**	**4-x**	**5-x**
1	310.45	127.57	97.53	97.53	97.53
2	194.45	60.92	47.74	47.74	47.74
3	144.19	41.07	31.58	31.58	31.58
4	104.88	28.21	21.25	21.25	21.25
5	59.04	15.06	11.09	11.09	11.09

Tabla 6.5 Rigideces de entrepiso de los muros paralelos al eje x (t/cm) (continúa)

Entrepis	**6-x**	**7-x**	**8-x**	**9-x**	**Σ**
1	97.53	97.53	127.57	310.45	1363.69
2	47.74	47.74	60.92	194.45	749.62
3	31.58	31.58	41.07	144.19	528.42
4	21.25	21.25	28.21	104.88	372.46
5	11.09	11.09	15.06	59.04	203.65

Tabla 6.6 Rigideces de entrepiso de los muros paralelos al eje y (t/cm)				
Nivel	**1-y**	**2-y**	**3-y**	**Σ**
1	249.88	114.32	151.08	515.28
2	125.33	53.84	73.98	253.15
3	87.23	35.96	50.66	173.85
4	63.14	25.12	33.02	121.28
5	33.86	13.04	19.03	65.93

6.3.2.1 Matriz de rigideces de los modelos unidimensionales

Al establecer las ecuaciones de equilibrio de los modelos estructurales mostrados en las Fig 6.4 se obtiene la siguiente matriz de rigideces.

$$\vec{K} = \begin{bmatrix} k_1 + k_2 & -k_2 & 0 & 0 & 0 \\ -k_1 & K_2 + k_3 & -k_3 & 0 & 0 \\ 0 & -k_3 & k_3 + k_4 & -k_4 & 0 \\ 0 & 0 & -k_4 & k_4 + k_5 & -k_5 \\ 0 & 0 & 0 & -k_5 & k_5 \end{bmatrix} \tag{6.8}$$

6.3.2.2 Matriz de masas de los modelos unidimensionales

Al establecer las ecuaciones de equilibrio de los modelos estructurales mostrados en las Fig 6.4 se obtiene la siguiente matriz de masas (concentradas).

$$\vec{M} = \frac{1}{g} \begin{bmatrix} W_1 & 0 & 0 & 0 & 0 \\ 0 & W_2 & 0 & 0 & 0 \\ 0 & 0 & W_3 & 0 & 0 \\ 0 & 0 & 0 & W_4 & 0 \\ 0 & 0 & 0 & o & W_5 \end{bmatrix} \tag{6.9}$$

6.3.2.3 Eigenvalores y eigenvectores

Al sustituír los valores de la tabla 6.1, 6.5 y 6.6 en las ecuaciones 6.8 y 6.9, para cada uno de los modelos estructurales asociados a las dos direcciones ortogonales, y resolver los correspondiente problemas de valores característicos, se obtienen los eigenvectores (formas modales) mostradas en la Fig 6.5.

Los valores de los períodos, frecuencias naturales de vibración y valores característicos correspondientes a los eigenvectores de la Fig 6.5 se se presentan en la tabla 6.7

Tabla 6.7 Períodos y frecuencias naturales de vibración de los modelos estructurales del edificio

Modo	Modelo estructural, eje y			Modelo estructural, eje x		
	T_{ix} (s)	$_{iy}$ (rad/s)	$_{iy}^{2}$ (rad/s)^2	T_{ix} (s)	$_{ix}$ (rad/s)	$_{ix}^{2}$ (rad/s)^2
1	.4719	13.31	177.28	.2735	22.97	527.77
2	.2006	31.32	981.06	.1158	54.26	2944.03
3	.1302	48.26	2328.83	.0752	83.55	6981.10
4	.0945	66.49	4420.75	.0548	114.66	13146.15
5	.0676	92.95	8639.06	.0401	156.69	24551.10

6.3.3 Respuesta espectral de desplazamientos de cada modo para el modelo estructural paralelo a eje y

Con base en los incisos 4.1.4.3 y 4.1.4.4 la respuesta espectral desplazamientos de cada modo se obtiene con las Ec 4.25 y 4.27, y de acuerdo con el inciso 4.1.3.3 el coeficiente de participación se obtiene con la Ec 4.22e, que se reproducen a continuación.

$$c_i = \frac{\sum_{k=1}^{N} m_k r_k^i}{\sum_{k=1}^{N} m_k (r_k^i)^2} \tag{4.22e}$$

$$y^i_{máx} = c_i \frac{A_i}{\omega_i^2} \tag{4.25}$$

$$\vec{u}^i_{máx} = \vec{r}^{\,i} y^i_{máx} \tag{4.27}$$

6.3.3.1 Primer modo

Las operaciones de las Ec 4.25 y 4.27 se presentan en la tabla 6.8. La columna 2 de dicha tabla se obtiene de la columna 2 de la tabla 6.1.

Tabla 6.8 Respuesta espectral de desplazamientos: Primer modo

Nivel k-ésimo	**m_k ts^2/cm**	**r_k^1**	**$m_k r_k 1$ ts^2/cm**	**$m_k(r_k^1)^2$ ts^2/cm**	**$u_{kmáx}^1$ cm**
1	0.106	1.0000	0.1060	0.1060	0.1127
2	0.106	2.9613	0.3139	0.9295	0.3337
3	0.106	5.4973	0.5827	3.2034	0.6195
4	0.106	8.2805	0.8777	7.2681	0.9332
5	0.093	11.0399	1.0267	11.3348	1.2442
Σ			2.9070	22.8418	

Con base en las columnas 4 y 5 se obtiene el valor del coeficiente de participación del modo 1, que resulta ser.

$$c_1 = \frac{2.9070}{22.8418} = 0.1273 \tag{6.10}$$

La ordenada del espectro de aceleraciones de diseño del primer modo de vibración, de acuerdo con el incisos 3.6 y 6.1.6, es.

$$\begin{aligned} T_a &= 0.2 < T_1 = 0.4719 < T_b = 0.6 \\ A_1 &= ag = cg = 0.16*981 = 156.96 \;\; cm/s^2 \end{aligned} \tag{6.11}$$

La respuesta espectral de las ecuaciones de equilibrio desacopladas para el primer modo, de acuerdo con la Ec 4.25 resulta ser.

$$y^1_{máx} = c_1 \frac{A_1}{\omega_1^2} = 0.1273 \frac{156.96}{177.28} = 0.1127 \quad cm \qquad (6.12)$$

La sexta columna de la tabla 6.8 es la expresión de la Ec 4.27.

6.3.3.2 Segundo modo

Las operaciones de las Ec 4.25 y 4.27 se presentan en la tabla 6.9. La columna 2 de dicha tabla se obtiene de la columna 2 de la tabla 6.1.

Tabla 6.9 Respuesta espectral de desplazamientos: Segundo modo

Nivel k-ésimo	m_k ts^2/cm	r_k^2	$m_k r_k 2$ ts^2/cm	$m_k(r_k^2)^2$ ts^2/cm	$u_{kmáx}^2$ cm
1	0.106	1.0000	0.1060	0.1060	0.0208
2	0.106	2.6245	0.2782	0.7301	0.0546
3	0.106	3.4198	0.3625	1.2397	0.0711
4	0.106	1.6282	0.1724	0.2803	0.0339
5	0.093	-4.2387	-0.3942	1.6709	-0.0882
Σ			0.5249	4.0270	

Con base en las columnas 4 y 5 se obtiene el valor del coeficiente de participación del modo 2, que resulta ser.

$$c_2 = \frac{0.5249}{4.0270} = 0.1303 \qquad (6.13)$$

La ordenada del espectro de aceleraciones de diseño del segundo modo de vibración, de acuerdo con el incisos 3.6 y 6.1.6, es.

$$\begin{aligned} T_a &= 0.2 < T_2 = 0.2006 < T_b = 0.6 \\ A_2 &= ag = cg = 0.16*981 = 156.96 \quad cm/s^2 \end{aligned} \qquad (6.14)$$

La respuesta espectral de las ecuaciones de equilibrio desacopladas para el segundo modo, de acuerdo con la Ec 4.25 resulta ser.

$$y^2_{m\acute{a}x} = c_2 \frac{A_2}{\omega_2^2} = 0.1303 \frac{156.96}{981.06} = 0.0208 \ cm \qquad (6.15)$$

La sexta columna de la tabla 6.9 es la expresión de la Ec 4.27.

6.3.3.3 Tercer modo

Las operaciones de las Ec 4.25 y 4.27 se presentan en la tabla 6.10. La columna 2 de dicha tabla se obtiene de la columna 2 de la tabla 6.1.

Tabla 6.10 Respuesta espectral de desplazamientos: Tercer modo

Nivel k-ésimo	m_k ts^2/cm	r_k^3	$m_k r_k 3$ ts^2/cm	$m_k(r_k^3)^2$ ts^2/cm	$u_{km\acute{a}x}^3$ cm
1	0.106	1.0000	0.1060	0.1060	0.0074
2	0.106	2.0606	0.2184	0.4501	0.0152
3	0.106	0.6797	0.0720	0.0490	0.0050
4	0.106	-2.6831	-0.2844	0.7631	-0.0197
5	0.093	1.1754	0.1073	0.1285	0.0086
Σ			0.2213	1.4967	

Con base en las columnas 4 y 5 se obtiene el valor del coeficiente de participación del modo 3, que resulta ser.

$$c_3 = \frac{0.2213}{1.4967} = 0.1479 \qquad (6.16)$$

La ordenada del espectro de aceleraciones de diseño del tercer modo de vibración, de acuerdo con el incisos 3.6 y 6.1.6, es.

$$T_3 = 0.1302 < T_a = 0.2$$

$$A_3 = ag = g(1 + 3\frac{T_3}{T_a})\frac{c}{4} \tag{6.17}$$

$$= 981(1 + 3\frac{0.1302}{0.2})\frac{0.16}{4} = 115.88 \quad cm/s^2$$

La respuesta espectral de las ecuaciones de equilibrio desacopladas para el tercer modo, de acuerdo con la Ec 4.25 resulta ser.

$$y^3_{máx} = c_3\frac{A_3}{\omega_3^2} = 0.2213\frac{115.88}{2328.83} = 0.0110 \quad cm \tag{6.18}$$

La sexta columna de la tabla 6.10 es la expresión de la Ec 4.27.

6.3.3.4 Cuarto modo

Las operaciones de las Ec 4.25 y 4.27 se presentan en la tabla 6.11. La columna 2 de dicha tabla se obtiene de la columna 2 de la tabla 6.1.

Tabla 6.11 Respuesta espectral de desplazamientos: Cuarto modo

Nivel k-ésimo	**m_k ts^2/cm**	**r_k^4**	**$m_k r_k 4$ ts^2/cm**	**$m_k(r_k^4)^2$ ts^2/cm**	**$u_{kmáx}^4$ cm**
1	0.106	1.0000	0.1060	0.1060	0.0039
2	0.106	1.1851	0.1256	0.1489	0.0046
3	0.106	-1.7383	-0.1843	0.3203	-0.0068
4	0.106	0.7849	0.0832	0.0653	0.0031
5	0.093	-0.1501	-0.0140	0.0021	-0.0006
Σ			0.1165	0.6426	

Con base en las columnas 4 y 5 se obtiene el valor del coeficiente de participación del modo 4, que resulta ser.

$$C_4 = \frac{0.1165}{0.6426} = 0.1813 \tag{6.19}$$

La ordenada del espectro de aceleraciones de diseño del cuarto modo de vibración, de acuerdo con el incisos 3.6 y 6.1.6, es.

$$\begin{aligned} T_4 &= 0.0945 < T_a = 0.2 \\ A_4 &= ag = g(1 + 3\frac{T_3}{T_a})\frac{c}{4} \\ &= 981(1 + 3\frac{0.0945}{0.2})\frac{0.16}{4} = 94.86 \ cm/s^2 \end{aligned} \tag{6.20}$$

La respuesta espectral de las ecuaciones de equilibrio desacopladas para el cuarto modo, de acuerdo con la Ec 4.25 resulta ser.

$$y^4_{máx} = C_4\frac{A_4}{\omega_4^2} = 0.1813\frac{94.86}{4420.75} = 0.00389 \ cm \tag{6.21}$$

La sexta columna de la tabla 6.11 es la expresión de la Ec 4.27.

6.3.3.5 Quinto modo

Las operaciones de las Ec 4.25 y 4.27 se presentan en la tabla 6.12. La columna 2 de dicha tabla se obtiene de la columna 2 de la tabla 6.1.

Tabla 6.12 Respuesta espectral de desplazamientos: Quinto modo

Nivel k-ésimo	m_k ts^2/cm	r_k^5	$m_k r_k^5$ ts^2/cm	$m_k(r_k^5)^2$ ts^2/cm	$u_{kmáx}^5$ cm
1	0.106	1.0000	0.1060	0.1060	0.0038
2	0.106	-0.5787	-0.0613	0.0355	-0.0022
3	0.106	0.1678	0.0178	0.0030	0.0006
4	0.106	-0.0282	-0.0030	0.0001	-0.0001
5	0.093	0.0025	-0.0002	0.0000	0.0000
Σ			0.0597	0.1446	

Con base en las columnas 4 y 5 se obtiene el valor del coeficiente de participación del modo 5, que resulta ser.

$$c_5 = \frac{0.0577}{0.1446} = 0.4129 \tag{6.22}$$

La ordenada del espectro de aceleraciones de diseño del quinto modo de vibración, de acuerdo con el incisos 3.6 y 6.1.6, es.

$$\begin{aligned} T_5 &= 0.0676 < T_a = 0.2 \\ A_5 &= ag = g\left(1 + 3\frac{T_3}{T_a}\right)\frac{c}{4} \\ &= 981\left(1 + 3\frac{0.0676}{0.2}\right)\frac{0.16}{4} = 79.03 \quad cm/s^2 \end{aligned} \tag{6.23}$$

La respuesta espectral de las ecuaciones de equilibrio desacopladas para el quinto modo, de acuerdo con la Ec 4.25 resulta ser.

$$y_{máx}^5 = c_5\frac{A_5}{\omega_5^2} = 0.4129\frac{79.03}{8639.06} = 0.003777 \quad cm \tag{6.24}$$

La sexta columna de la tabla 6.12 es la expresión de la Ec 4.27.

6.3.4 Respuesta espectral de fuerzas cortantes de cada modo para el modelo estructural paralelo al eje y

Con base en la respuesta espectral de desplazamientos de cada modo (cuantificados en la columna 6 de las tablas 6.8 a 6.12, que se repiten en la columna 2 de las tablas 6.13 a 6.17) se pueden cuantificar las fuerzas cortantes correspondientes, al utilizar los conceptos relacionados con la definición de rigidez de entrepiso (Ec 2.5 y 2.6), reproducidos en la forma en que se utilizan.

$$V_k = k_k \Delta u_k \tag{2.5}$$

$$\Delta u_k = u_k - u_{k-1} \tag{2.6}$$

Los valores de las rigideces de entrepiso para el modelo estructural paralelo al eje y se muestran en la columna 2 de la tabla 6.2 o bien en la columna 12 de la tabla 6.5, y se repiten sistemáticamente en la columna 3 de las tablas 6.13 a 6.17.

En la revisión del cumplimiento de las condiciones de regularidad del edificio respecto a la rigidez al corte (inciso 3.4), la relación de rigideces entre el primer y segundo entrepisos es igual a 2.035. Aunque excede del 100 por ciento (103.5), se considera que la rigidez del primer entrepiso está sobrevaluada por la condición de frontera de empotramiento. Por tanto, el edificio es regular y los factores reductivos Q' no sufren reducciones adicionales.

6.3.4.1 Primer modo

Las operaciones de las Ec 2.6 y 2.5 se presentan en las columnas 4 y 5 de la tabla 6.13.

Tabla 6.13 Respuesta espectral de cortantes: Primer modo

Nivel/ Entrepis	$u_{kmáx}^{1}$ cm	k_{ky} t/cm	$u_{kmáx}^{1}$ cm	$V_{kmáx}^{1}$ t	$V_{kmáxr}1$ t
1	0.1127	515.28	0.1127	58.12	38.75
2	0.3337	253.15	0.2210	55.95	37.30
3	0.6195	173.85	0.2858	49.68	33.12
4	0.9332	121.28	0.3137	38.05	25.37
5	1.2442	65.93	0.3110	20.50	13.67

La sexta columna representa los valores de la fuerza cortante reducida al dividir los valores de la quinta columna entre el factor reductivo Q'_{1y}, que resulta ser.

$$\begin{aligned} T_{1y} &= 0.4719 > T_a = 0.2 \\ Q'_{1y} &= Q_y = 1.500 \end{aligned} \tag{6.25}$$

6.3.4.2 Segundo modo

Las operaciones de las Ec 2.6 y 2.5 se presentan en las columnas 4 y 5 de la tabla 6.14.

Tabla 6.14 Respuesta espectral de cortantes: Segundo modo					
Nivel/ Entrepis	$u_{kmáx}^2$ cm	k_{ky} t/cm	$u_{kmáx}^2$ cm	$V_{kmáx}^2$ t	$V_{kmáxr}2$ t
1	0.0208	515.28	0.0208	10.72	7.15
2	0.0546	253.15	0.0338	8.56	5.71
3	0.0711	173.85	0.0165	2.87	1.91
4	0.0339	121.28	-0.0372	-4.51	-3.01
5	-0.0882	65.93	-0.1221	-8.05	-5.37

La sexta columna representa los valores de la fuerza cortante reducida al dividir los valores de la quinta columna entre el factor reductivo Q'_{2y}, que resulta ser.

$$\begin{aligned} T_{2y} &= 0.2006 > T_a = 0.2 \\ Q'_{2y} &= Q_y = 1.500 \end{aligned} \qquad (6.26)$$

6.3.4.3 Tercer modo

Las operaciones de las Ec 2.6 y 2.5 se presentan en las columnas 4 y 5 de la tabla 6.15.

Tabla 6.15 Respuesta espectral de cortantes: Tercer modo					
Nivel/ Entrepis	$u_{kmáx}^3$ cm	k_{ky} t/cm	$u_{kmáx}^3$ cm	$V_{kmáx}^3$ t	$V_{kmáxr}3$ t
1	0.0074	515.28	0.0074	3.81	2.87
2	0.0152	253.15	0.0078	1.97	1.49
3	0.0050	173.85	-0.0102	-1.77	-1.33
4	-0.0197	121.28	-0.0247	-3.00	-2.26
5	0.0086	65.93	0.0283	1.87	1.41

La sexta columna representa los valores de la fuerza cortante reducida al dividir los valores de la quinta columna entre el factor reductivo Q'_{3y}, que resulta ser.

$$T_{3y} = 0.1302 < T_a = 0.2$$

$$Q'_{3y} = 1 + \frac{T_{3y}}{T_a}(Q_y - 1) = 1.326 \qquad (6.27)$$

6.3.4.4 Cuarto modo

Las operaciones de las Ec 2.6 y 2.5 se presentan en las columnas 4 y 5 de la tabla 6.16.

Tabla 6.16 Respuesta espectral de cortantes: Cuarto modo

Nivel/ Entrepis	$u_{kmáx}^4$ cm	k_{ky} t/cm	$u_{kmáx}^4$ cm	$V_{kmáx}^4$ t	$V_{kmáxr}^4$ t
1	0.0039	515.28	0.0039	2.00	1.62
2	0.0046	253.15	0.0007	0.18	0.15
3	-0.0068	173.85	-0.0114	-1.98	-1.60
4	0.0031	121.28	0.0099	1.20	0.97
5	-0.0006	65.93	-0.0037	-0.24	-0.19

La sexta columna representa los valores de la fuerza cortante reducida al dividir los valores de la quinta columna entre el factor reductivo Q'_{4y}, que resulta ser.

$$T_{4y} = 0.0945 < T_a = 0.2$$

$$Q'_{4y} = 1 + \frac{T_{4y}}{T_a}(Q_y - 1) = 1.236 \qquad (6.28)$$

6.3.4.5 Quinto modo

Las operaciones de las Ec 2.6 y 2.5 se presentan en las columnas 4 y 5 de la tabla 6.17.

Tabla 6.17 Respuesta espectral de cortantes: Quinto modo					
Nivel/ Entrepis	$u_{kmáx}^5$ cm	k_{ky} t/cm	$u_{kmáx}^5$ cm	$V_{kmáx}^5$ t	$V_{kmáxr}^5$ t
1	0.0038	515.28	0.0038	1.94	1.66
2	-0.0022	253.15	-0.0060	-1.52	-1.3
3	0.0006	173.85	0.0028	0.49	0.42
4	-0.0001	121.28	-0.0007	-0.08	-0.07
5	0.0000	65.93	0.0001	0.01	0.01

La sexta columna representa los valores de la fuerza cortante reducida al dividir los valores de la quinta columna entre el factor reductivo Q'_{5y}, que resulta ser.

$$
\begin{aligned}
T_{5y} &= 0.0676 < T_a = 0.2 \\
Q'_{5y} &= 1 + \frac{T_{5y}}{T_a}(Q_y - 1) = 1.169
\end{aligned}
\qquad (6.29)
$$

6.3.5 Respuesta total para el modelo estructural paralelo al eje y

Conocidos los elementos cinemáticos (inciso 6.3.3) y los elementos mecánicos (inciso 6.2.4) del modelo estructural en estudio para cada modo de vibración, se procede a determinar la respuesta total de dicho modelo estructural.

Las NTC para diseño por sismo del RCDF87 establecen que debe incluírse el efecto de todos los modos naturales de vibración con período mayor o igual a 0.4 s, pero en ningún caso se pueden considerar menos que los tres primeros modos de traslación en cada dirección de análisis.

Las NTC para diseño por sismo del RCDF87 recomienda utilizar el método de la raíz cuadrada de la suma de los cuadrados (SRSS), para calcular la respuesta total, siempre que los períodos de los modos naturales en cuestión difieran al menos 10% entre si, que es el caso. el método SRSS se indica mediante la Ec 4.29, que se reproduce a continuación.

$$S = \sqrt{\sum_{i=1}^{N} S_i^2} \qquad (4.29)$$

6.3.5.1 Respuesta total de desplazamientos

En la tabla 6.18 se resumen las operaciones indicadas por la Ec 4.29 para los vectores de desplazamientos máximos de cada modo mostrados en la columna 6 de las tablas 6.8 a 6.12. En la columna 2 se muestra la combinación de un solo modo (el primero), en la columna 3 la combinación de los dos primeros, y asi sucecivamente.

El primer elemento de cada casillero representa el componente de desplazamiento total mientras que el segundo elemento representa el cociente de ese desplazamiento entre el desplazamiento total obtenido con la combinación de todos los modos del modelo estructural, dados por la columna 6.

Tabla 6.18 Respuesta total de desplazamientos (cm)

Nivel	1 modo	2 modos	3 modos	4 modos	5 modos
1	0.1127 0.98	0.1146 0.99	0.1148 1.00	0.1149 1.00	0.1150 1.00
2	0.3337 0.97	0.3381 1.00	0.3385 1.00	0.3385 1.00	0.3385 1.00
3	0.6195 0.99	0.6236 1.00	0.6236 1.00	0.6236 1.00	0.6236 1.00
4	0.9332 1.00	0.9338 1.00	0.9349 1.00	0.9340 1.00	0.9340 1.00
5	1.2442 1.00	1.2473 1.00	1.2474 1.00	1.2474 1.00	1.2474 1.00

6.3.5.2 Respuesta total de fuerzas cortantes

En la tabla 6.19 se resumen las operaciones indicadas por la Ec 4.29 para los vectores de fuerzas cortantes máximos de cada modo mostrados en la columna 6 de las tablas 6.13 a 6.17. El ordenamiento de esta tabla es enteramente similar al de la tabla 6.18.

Tabla 6.19	**Respuesta total de fuerzas cortantes (t)**					**V_t Escala**
Entrepi	**1 modo**	**2 modos**	**3 modos**	**4 modos**	**5 modos**	
1	37.75 0.98	38.42 0.99	38.53 1.00	38.56 1.00	38.60 1.00	43.28
2	37.30 0.99	37.73 1.00	37.76 1.00	37.76 1.00	37.79 1.00	42.37
3	33.12 0.99	33.18 1.00	33.20 1.00	33.24 1.00	33.24 1.00	37.27
4	25.37 1.00	25.55 1.00	25.65 1.00	25.67 1.00	25.67 1.00	28.78
5	13.67 0.93	14.69 0.99	14.75 1.00	14.76 1.00	14.56 1.00	16.55

6.3.5.3 Revisión por cortante basal

Las NTC para diseño por sismo del RCDF87 establecen que si con el método de análisis dinámico que se haya aplicado se encuentra que, en la dirección que se considera, la fuerza cortante basal calculada, V_0, debe ser tal que debe cumplir con la siguiente condición.

$$V_0 \geq 0.8a\frac{W_0}{Q'} = (0.8)(0.16)\frac{507.2}{1.5} = 43.28 \ t \qquad (6.30)$$

En caso de no cumplirse la condición anterior, Las fuerzas de diseño y los desplazamientos laterales correspondientes se deben incrementar en la proporción para que el cortante basal calculado, V_0, cumpla con la igualdad.

De acuerdo con la tabla 6.19, el cortante basal que proporciona el método dinámico es, V_0 = 38.60 t, por lo que las fuerzas cortantes que proporciona el método dinámico (columna 6 de la tabla 6.19) se deben multiplicar por el coeficiente, 43.28/38.6 = 1.12. El escalamiento se indica en la columna 7 de la tabla 6.19.

6.3.6 Comparación de las fuerzas cortantes obtenidas con los métodos estático y dinámico

A fin de tener una idea comparativa de los valores de las fuerzas cortantes que cada método proporciona se construye la tabla 6.20 donde se establecen tales comparaciones.

Tabla 6.20 Comparación de fuerzas cortantes sísmicas

Entrepiso	V_{est} (t)	V_{din} (t)	V_{est}/V_{din}
1	54.09	43.28	1.25
2	50.33	42.37	1.19
3	42.81	37.27	1.29
4	31.53	28.78	1.10
5	16.48	16.55	1.00

6.4 Fuerzas sísmicas en los elementos estructurales de la edificación

6.4.1 Resumen de las ecuaciones utilizadas

En el inciso 5.2 se presenta el procedimiento para cuantificar las fuerzas sismicas para el modelo estructural que utiliza el concepto de rigideces de entrepiso. Las ecuaciones que se utilizan se reproducen a continuación.

6.4.1.1 Coordenadas del centro de torsión

$$x_t = \frac{\sum_{j=1}^{NX} x_j k_{jy}}{\sum_{j=1}^{NX} k_{jy}} \qquad (5.10)$$

$$y_t = \frac{\sum_{i=1}^{NY} y_i k_{ix}}{\sum_{i=1}^{NY} k_{ix}} \tag{5.11}$$

6.4.1.2 Fuerzas cortantes directas

$$V_{jy}^d = \frac{k_{jy}}{\sum_{j=1}^{NX} k_{jy}} V_y \tag{5.4}$$

$$V_{ix}^d = \frac{k_{ix}}{\sum_{i=1}^{NY} k_{ix}} V_x \tag{5.8}$$

6.4.1.3 Excentricidades calculadas

$e_{sx} = |x_m - x_t|$

donde :

e_{sx} = Excentricidad de la fuerza Cortante V_y (5.12)

x_m = Abscisa del centro de masas

x_t = Abscisa del centro de torsión

$$e_{sy} = |y_m - y_t|$$

donde :

e_{sy} : Excentricidad de la fuerza Cortante V_x (5.13)

y_m : Ordenada del centro de masas

y_t : Ordenada del centro de torsión

6.4.1.4 Excentricidades de diseño

$$\begin{aligned} e_{dx} &= 1.5e_{sx} + 0.1b_x \\ e_{dx} &= e_{sx} - 0.1b_x \end{aligned} \tag{5.14}$$

b_x es la dimensión de la planta que se considera medida en la dirección de e_{sx} (perpendicular a la fuerza cortante V_y).

$$\begin{aligned} e_{dy} &= 1.5e_{sy} + 0.1b_y \\ e_{dy} &= e_{sy} - 0.1b_y \end{aligned} \tag{5.15}$$

b_y es la dimensión de la planta que se considera medida en la dirección de e_{sy} (perpendicular a la fuerza cortante V_x).

6.4.1.5 Fuerzas cortantes debidas a la torsión

$$\begin{aligned} M &= M_{ty} = e_{dx}V_y \\ &= M_{tx} = e_{dy}V_x \end{aligned} \tag{5.16}$$

$$V_{ix}^{t} = \frac{k_{ix}\overline{y}_i}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M \tag{5.22}$$

$$V_{jy}^{t} = \frac{k_{jy}\bar{x}_j}{\sum_{i=1}^{NY} k_{ix}\bar{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\bar{x}_j^2} M \qquad (5.23)$$

6.4.1.6 Fuerzas cortantes de diseño en los resortes (rigideces de entrepiso)

$$\begin{aligned} V_{ix} &= V_{ix}^{d} + V_{ix}^{t} \\ V_{jy} &= V_{jy}^{d} + V_{jy}^{t} \end{aligned} \qquad (5.24)$$

Las fuerzas cortantes que se utilizan son las obtenidas con el método estático, ya que con el método dinámico se obtuvieron para el modelo estructural paralelo a la dirección del eje y.

6.4.2 Diafragma del nivel 1

En la Fig 6.6 se muestra la geometría del diafragma del nivel 1 así como la distribución de las rigideces de entrepiso que llegan a dicho nivel y la posición del centro de masas. Con base en dicha figura y las ecuaciones resumidas del capítulo 5 se construyen las tabla 6.21 y 6.22

Con base en las columnas 3 y 4 de la tabla 6.21 y la Ec 5.11 se obtiene el siguiente valor de la ordenada del centro de torsión.

$$y_{1t} = \frac{1084134}{136369} = 7.95 \ m \qquad (6.31)$$

Con los elementos de la columna 3 de la tabla 6.21, la fuerza cortante correspondiente y la Ec 5.8 se obtienen los elementos de la columna 5 de dicha tabla.

Los elementos de la columna 6 de la tabla 6.21 se obtiene mediante la Ec 6.31 y la columna 2 de dicha tabla.

Tabla 6.21	Fuerzas sísmicas en las rigideces del Entrepiso 1, paralelas al eje x		
Eje i-x	**y_i (m)**	**k_{ix} (t/m)**	**$y_i k_{ix}$ (t)**
1-x	0.00	31045.00	0.00
2-x	2.85	12757.00	36357.00
3-x	4.20	9753.00	40963.00
4-x	6.60	9753.00	64370.00
5-x	7.95	9753.00	77536.00
6-x	9.30	9753.00	90703.00
7-x	11.70	9753.00	114110.00
8-x	13.05	12757.00	166479.00
9-x	15.90	31045.00	493616.00
Σ		136369.00	1084134.00

Tabla 6.21	Fuerzas sísmicas en las rigideces del Entrepiso 1, paralelas al eje x (cont)			
Eje i-x	V^{d}_{1ix} (t)	$\overline{y}_i$ (m)	$\overline{y}_i k_{ix}$ (t)	$\overline{y}_i^2 k_{ix}$ (tm)
1-x	12.31	-7.95	-246808.0	1962121.0
2-x	5.06	-5.10	-65061.0	331810.0
3-x	3.87	-3.75	-36574.0	137152.0
4-x	3.87	-1.35	-13167.0	17775.0
5-x	3.87	0.00	0.0	0.0
6-x	3.87	1.35	13167.0	17775.0
7-x	3.87	3.75	36574.0	137152.0
8-x	5.06	5.10	65061.0	331810.0
9-x	12.31	7.95	246808.0	1962121.0
Σ	54.09			4897715.0

A fin de cuantificar la abscisa del centro de torsión y las demás elementos de las restantes ecuaciones del capítulo 5 se construye la tabla 6.22, con base en la Fig 6.6

Con base en las columnas 3 y4 de la tabla 6.22 y la Ec 5.10 se obtiene el siguiente valor de la abscisa del centro de torsión.

$$x_{1t} = \frac{174921}{51528} = 3.40 \ m \tag{6.32}$$

Con los elementos de la columna 3 de la tabla 6.22, la fuerza cortante correspondiente y la Ec 5.4 se obtienen los elementos de la columna 5 de dicha tabla.

Los elementos de la columna 6 de la tabla 6.22 se obtiene mediante la Ec 6.32 y la columna 2 de dicha tabla.

Tabla 6.22 Fuerzas sísmicas en las rigideces del Entrepiso 1, paralelas al eje y

Eje j-y	x_j (m)	k_{jy} (t/m)	$x_j k_{jy}$ (t)
1-y	0.00	24988.0	0.0
2-y	4.20	11432.0	48014.0
3-y	8.40	15108.0	126907.0
Σ		51528.0	174921.0

Tabla 6.22 Fuerzas sísmicas en las rigideces del Entrepiso 1, paralelas al eje y (cont)

Eje j-y	V^d_{1jy} (t)	$\overline{x}_j$ (m)	$\overline{x}_j k_{jy}$ (t)	$\overline{x}_j^2 k_{jy}$ (tm)
1-y	26.23	-3.40	-84459.0	288861.0
2-y	12.00	0.80	9146.0	7316.0
3-y	15.86	5.00	75540.0	377700.0
Σ	54.09			673877.0

De acuerdo con las coordenadas del dentro de masas especificado en la Fig 6.6 y las Ec 6.31 y 6.32 se obtienen los siguientes valores de las excentricidades calculadas, de acuerdo con las Ec 5.12 y 5.13.

$$
\begin{aligned}
e_{1sx} &= |x_{1m} - x_{1t}| = |4.20 - 3.40| = 0.80\ m \\
e_{1sy} &= |y_{1m} - y_{1t}| = |7.95 - 7.95| = 0.00\ m
\end{aligned}
\tag{6.33}
$$

Con base en las Ec 6.33, 5.14 y 5.15 se obtienen las excentricidades de diseño correspondientes.

$$
\begin{aligned}
e_{1dx} &= 1.5e_{1sx} + 0.1b_x = 1.5(0.8) + 0.1(8.4) = 2.04\ m \\
e_{1dx} &= e_{1sx} - 0.1b_x = 0.8 - 0.1(8.4) = -0.04\ m
\end{aligned}
\tag{6.34}
$$

$$
\begin{aligned}
e_{1dy} &= 1.5e_{1sy} + 0.1b_y = 1.5(0.0) + 0.1(15.9) = 1.59\ m \\
e_{1dy} &= e_{1sy} - 0.1b_y = 0.0 - 0.1(15.9) = -1.59\ m
\end{aligned}
\tag{6.35}
$$

Con base en las Ec 6.34, 6.35 y 5.16 se obtiene el momento torsionante que se las fuerzas sísmicas le ocasionan al diafragma rígido del nivel 1.

$$
\begin{aligned}
M_{1ty} &= e_{1dx}V_{1y} = 2.04(54.09) = 110.34\ tm \\
&= e_{1dx}V_{1y} = 0.04(54.09) = 2.20\ tm
\end{aligned}
\tag{6.36}
$$

$$
\begin{aligned}
M_{1tx} &= e_{1dy}V_{1x} = 1.59(54.09) = 86.00\ tm \\
&= e_{1dy}V_{1x} = 1.59(54.09) = 86.00\ tm
\end{aligned}
\tag{6.37}
$$

De acuerdo con las Ec 5.22, 5.23 y la columna 8 de las tablas 6.21 y 6.22 se obtienen los siguientes coeficientes.

Las NTC para diseño por sismo, en su inciso 8.6, establece que de los dos momentos torsionantes de diseño en cada dirección (Ec 6.36 y 6.37) se debe tomar para cada marco o muro el que resulte mas desfavorable. Para cuantificar las fuerzas cortantes debidas a la torsión se utilizan las Ec 5.22 y 5.23, que de acuerdo con las columnas 8 de las tablas 6.21 y 6.22 y las Ec 6.36 y 6.37 resultan ser.

$$V_{1ix}^{ty} = \frac{k_{ix}\overline{y}_i}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M_{1ty} = \frac{110.34}{4897715 + 673877} k_{ix}\overline{y}_i \tag{6.38}$$

$$= 0.000019804 k_{ix}\overline{y}_i$$

$$V_{1jy}^{ty} = \frac{k_{jy}\overline{x}_j}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M_{1ty} = 0.000019804 k_{jy}\overline{x}_j \tag{6.39}$$

$$V_{1ix}^{tx} = \frac{k_{ix}\overline{y}_i}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M_{1tx} = \frac{86.00}{4897715 + 673877} k_{ix}\overline{y}_i \tag{6.40}$$

$$= 0.0000154354 k_{ix}\overline{y}_i$$

$$V_{1jy}^{tx} = \frac{k_{jy}\overline{x}_j}{\sum_{i=1}^{NY} k_{ix}\overline{y}_i^2 + \sum_{j=1}^{NX} k_{jy}\overline{x}_j^2} M_{1tx} = 0.0000154354 k_{jy}\overline{x}_j \tag{6.41}$$

En la Fig 6.7 se presentan las fuerzas cortantes, cuando el sismo de diseño actúa en uno de sus sentidos, dadas por las Ec 6.38 a 6.41 al utilizar los valores de la columna 7 de las tablas 6.21 y 6.22. Tales valores se presentan en las columnas 9 a 12 de las tablas 6.21 y 6.22, en donde se incluyen los dos sentidos en que puede actuar el sismo de diseño.

Las columnas 13 de las tablas 6.21 y 6.22 se cuantifican de acuerdo con las Ec 5.24, de tal manera que se obtenga la fuerza cortante mayor.

Tabla 6.21	Fuerzas sísmicas en las rigideces del Entrepiso 1, paralelas al eje x (cont)				
Eje i-x	V^{ty}_{1ix} (t)	$-V^{ty}_{1ix}$ (t)	V^{tx}_{1ix} (t)	$-V^{tx}_{1ix}$ (t)	V_{1ix} (t)
1-x	-4.89	4.89	3.81	-3.81	17.20
2-x	-1.29	1.29	1.00	-1.00	6.35
3-x	-0.72	0.72	0.56	-0.56	4.59
4-x	-0.26	0.26	0.20	-0.20	4.13
5-x	0.00	0.00	0.00	0.00	3.87
6-x	0.26	-0.26	-0.20	0.20	4.13
7-x	0.72	-0.72	-0.56	0.56	4.59
8-x	1.29	-1.29	-1.00	1.00	6.35
9-x	4.89	-4.89	-3.81	3.81	17.20
Σ					

Tabla 6.22	Fuerzas sísmicas en las rigideces del Entrepiso 1, paralelas al eje y (cont)				
Eje j-y	V^{ty}_{1jy} (t)	$-V^{ty}_{1jy}$ (t)	V^{tx}_{1jy} (t)	$-V^{tx}_{1jy}$ (t)	V_{1jy} (t)
1-y	-1.67	1.67	1.30	-1.30	27.90
2-y	0.18	-0.18	-0.14	0.14	12.18
3-y	1.50	-1.50	-1.17	1.17	17.36
Σ					

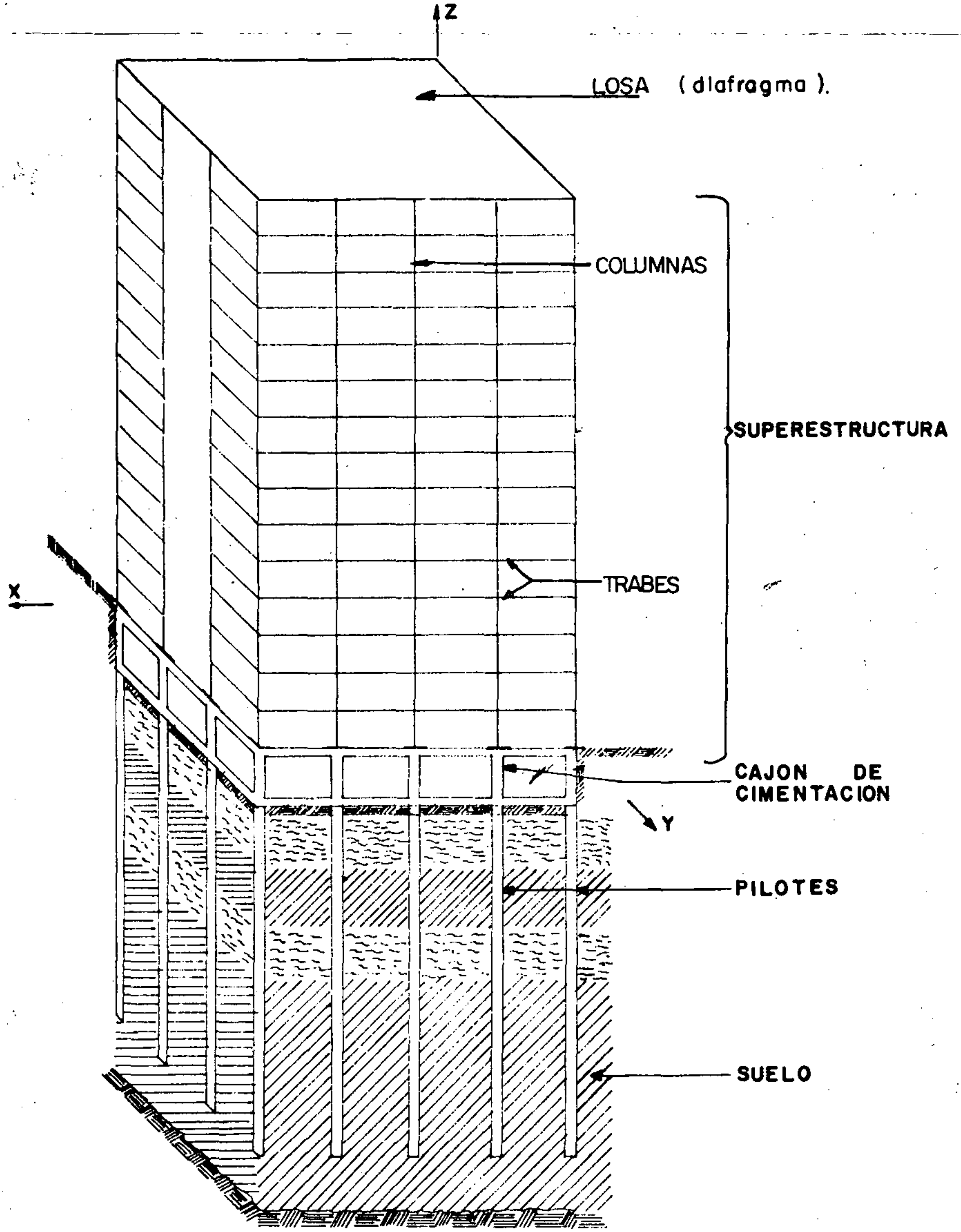


FIG 2.1 Representación esquemática de una edificación

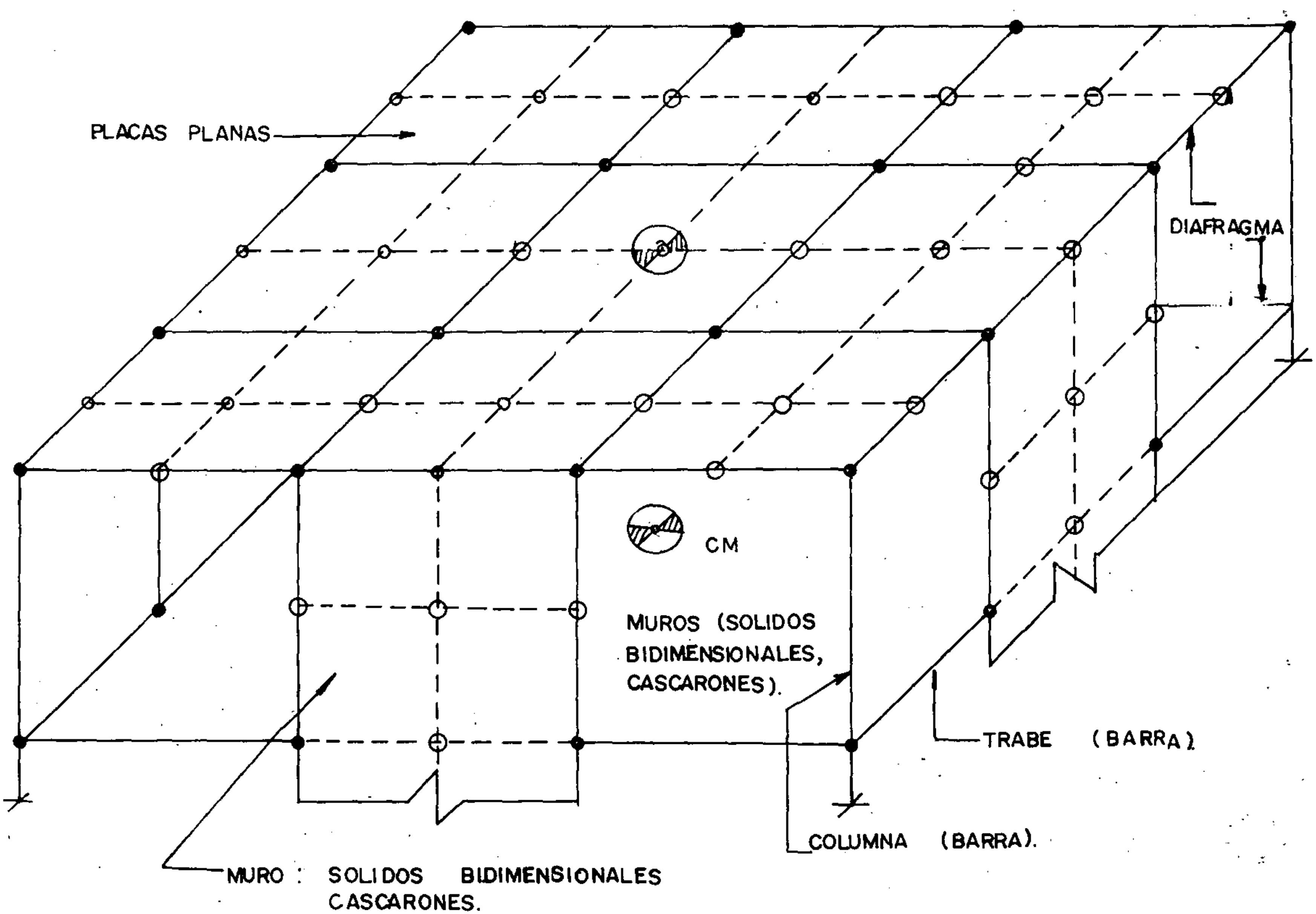


FIG 2.2 ELEMENTOS ESTRUCTURALES DE UNA EDIFICACION.

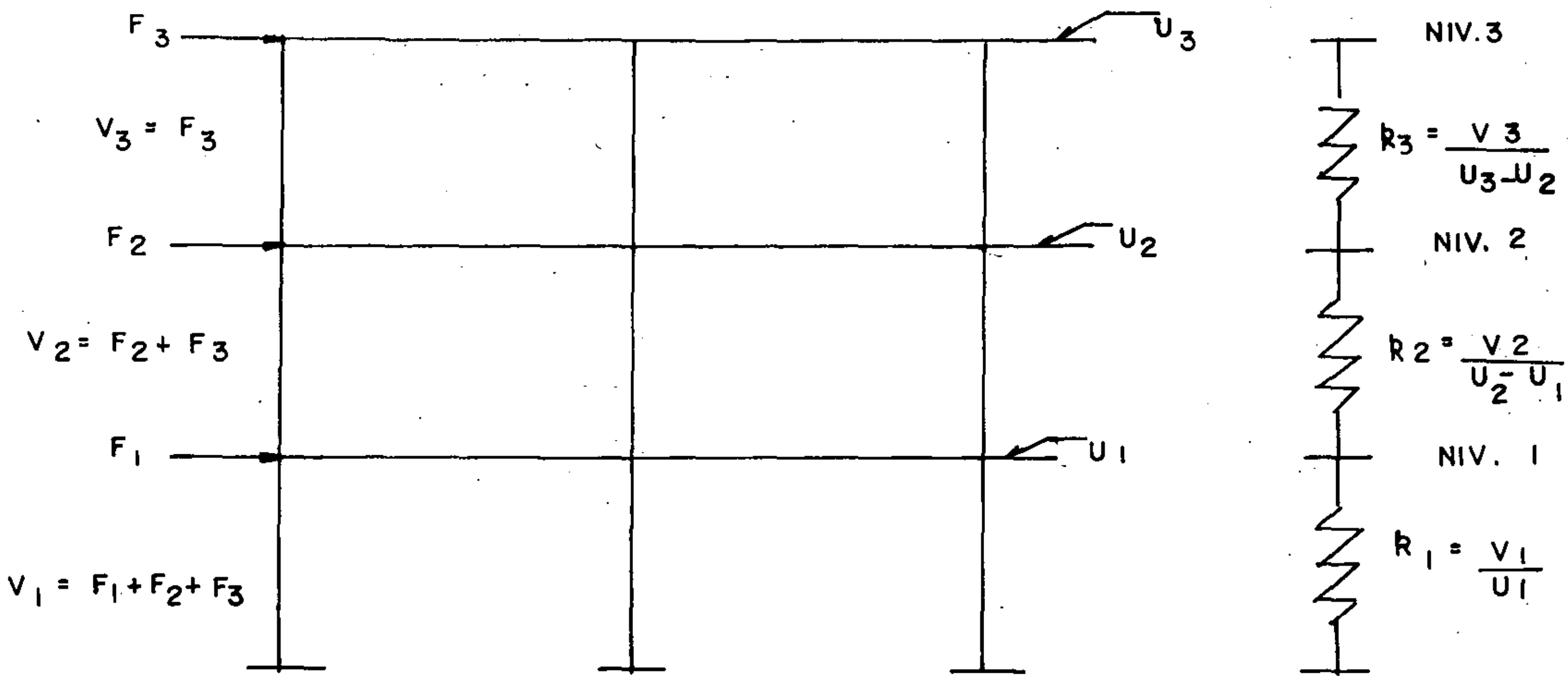


FIG 2.3 IDEALIZACION DE MARCOS PLANOS MEDIANTE RIGIDECES DE ENTREPISO.

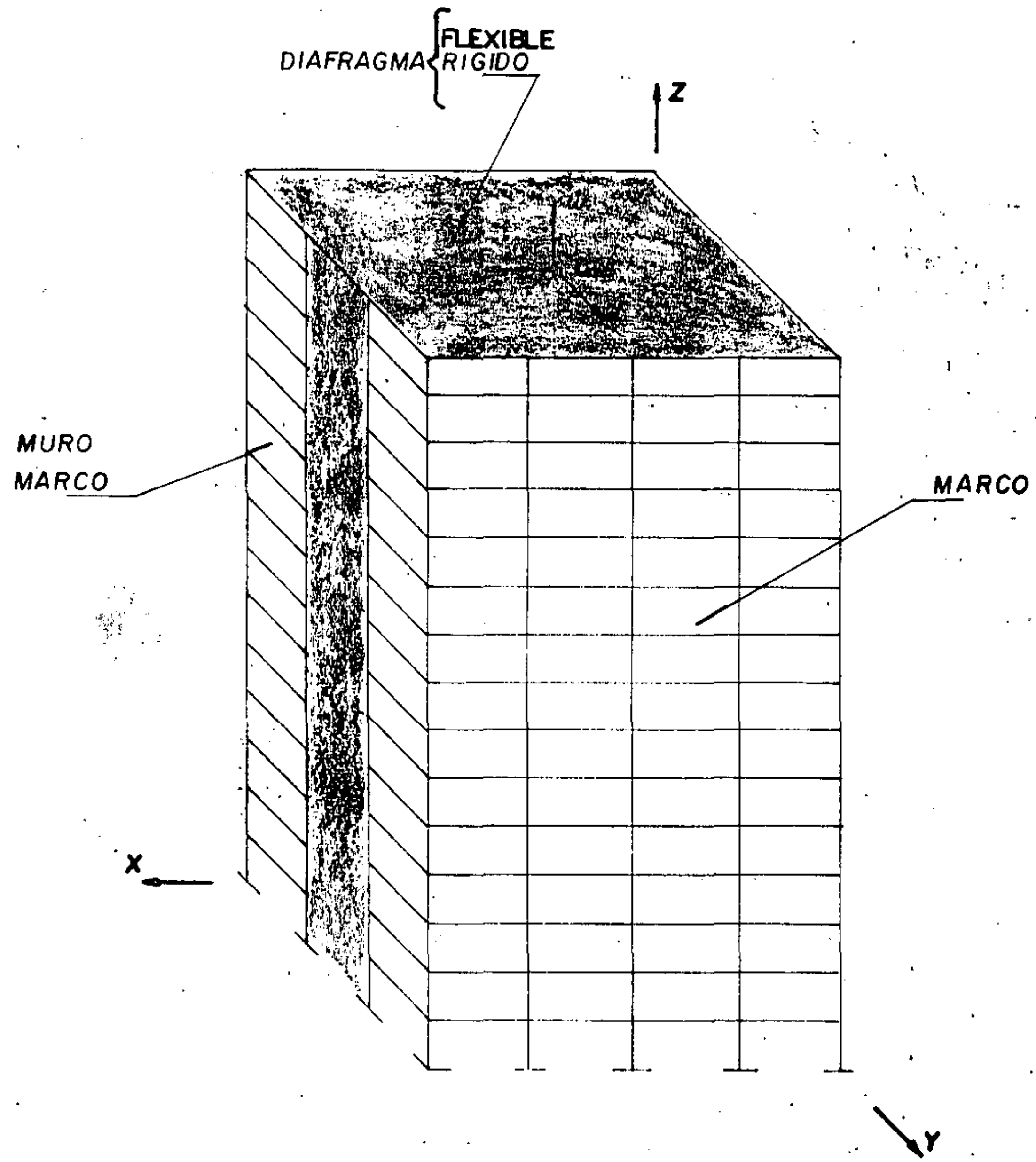


FIG 2.4 *Muros y Muromarcos Tridimensionales unidos con diafragmas*

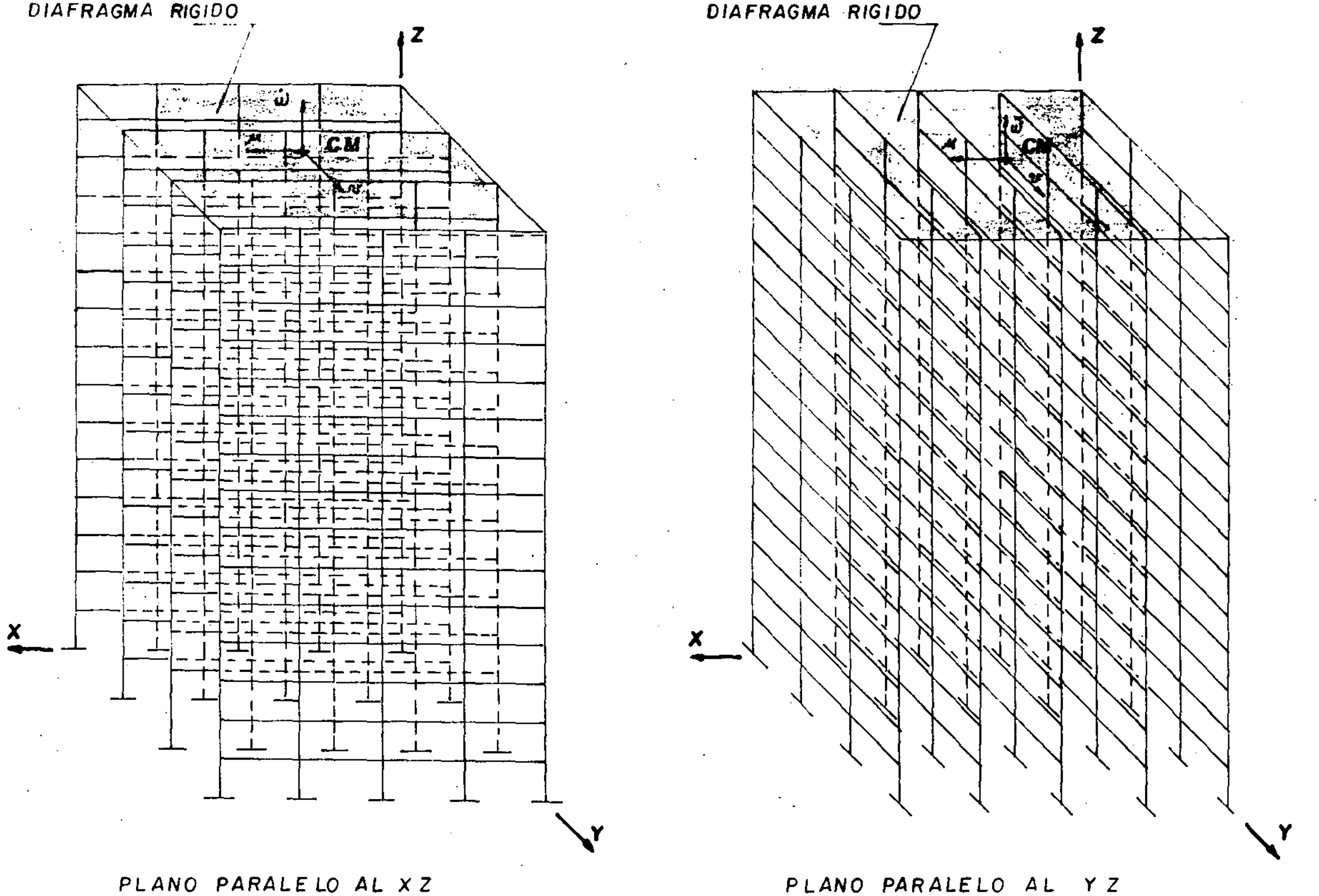


FIG 2.6 *Subestructuras formadas con marcos y muromarcos planos unidos con diafragmas rígidos*

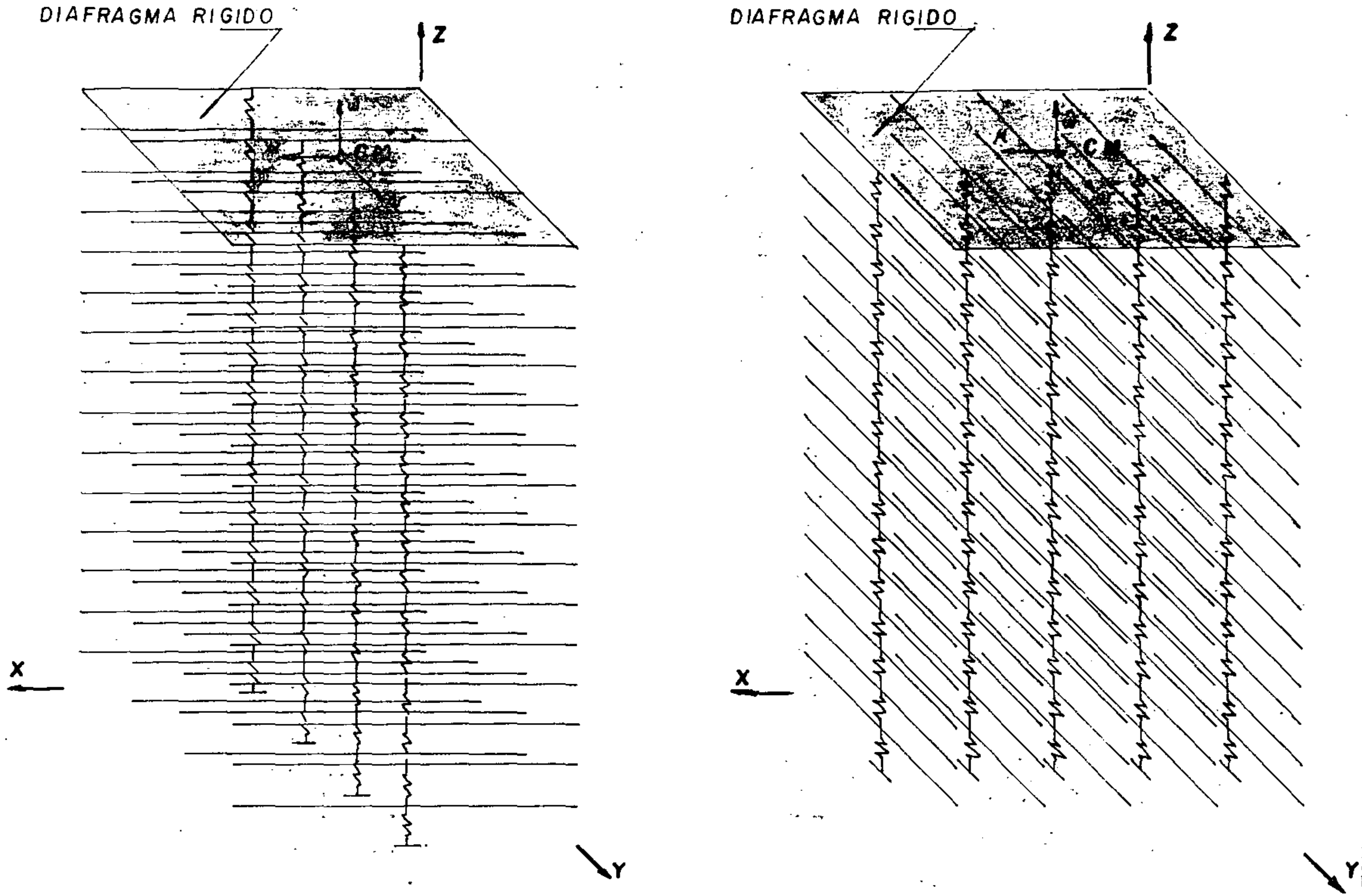


FIG 2.7 *Rigideces de entrepiso (resortes) unidas con diafragmas rígidos*

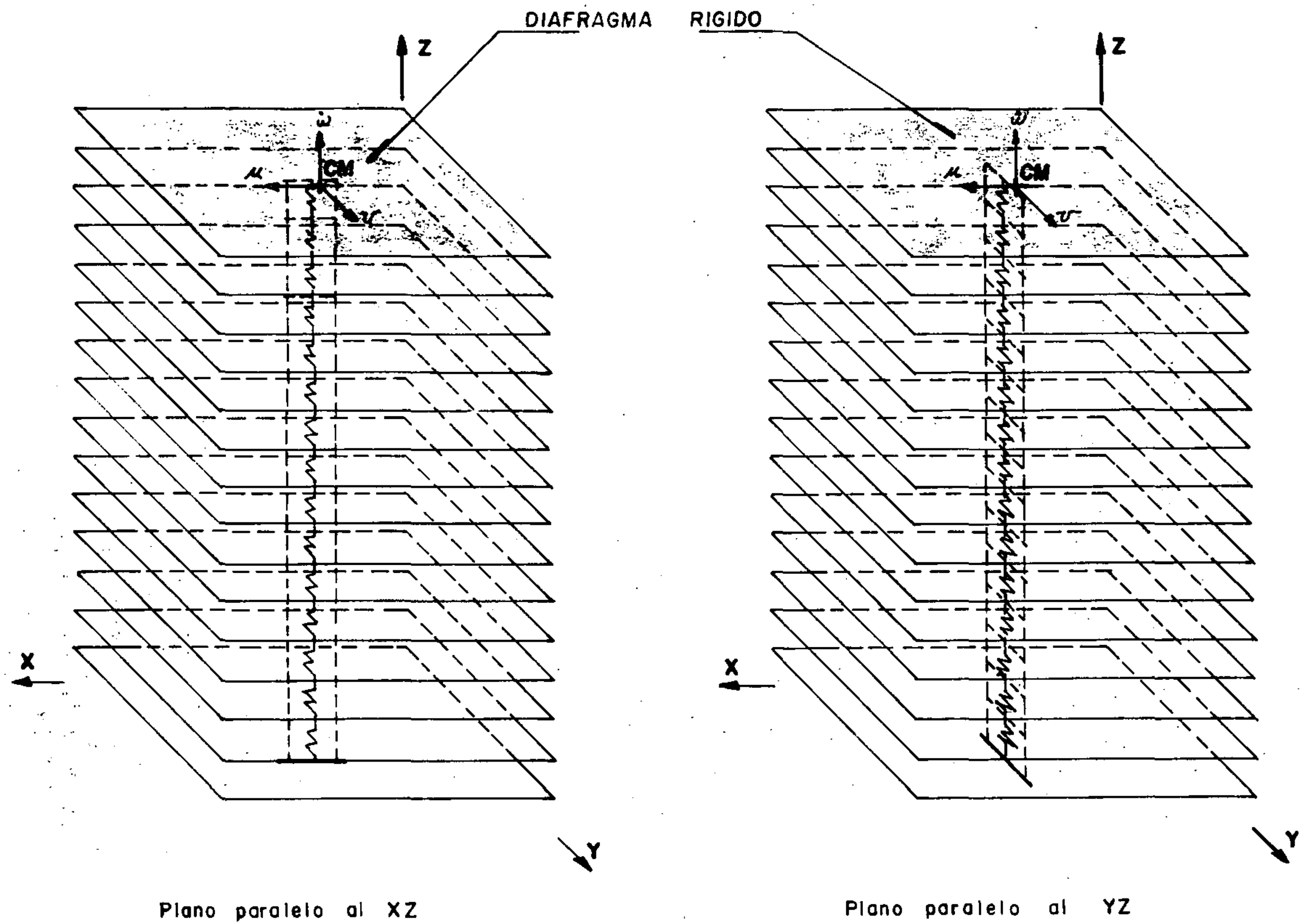


FIG 2.8 MODELOS UNIDIRECCIONALES INDEPENDIENTES FORMADOS CON LAS RIGIDECES DE ENTREPISO.

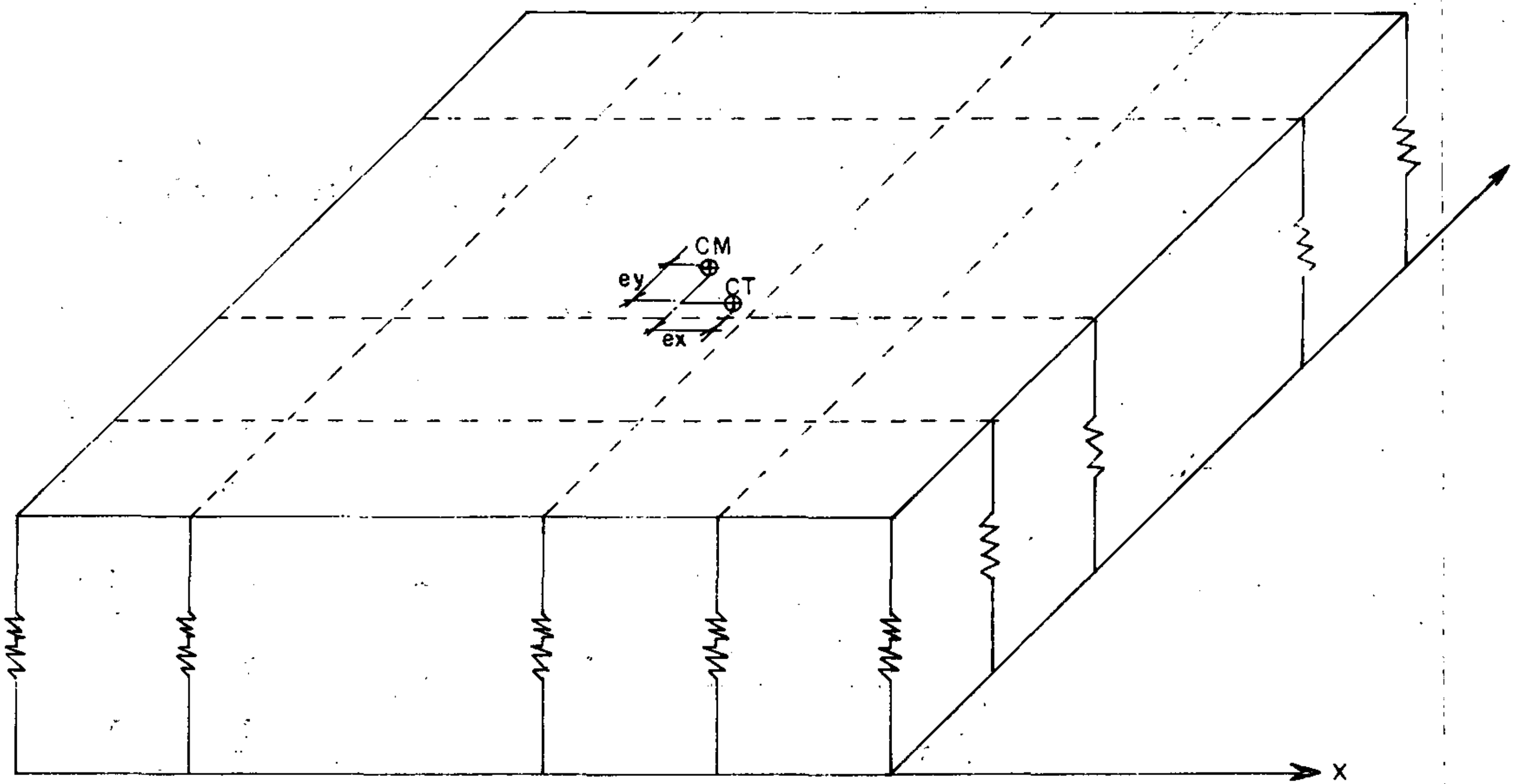
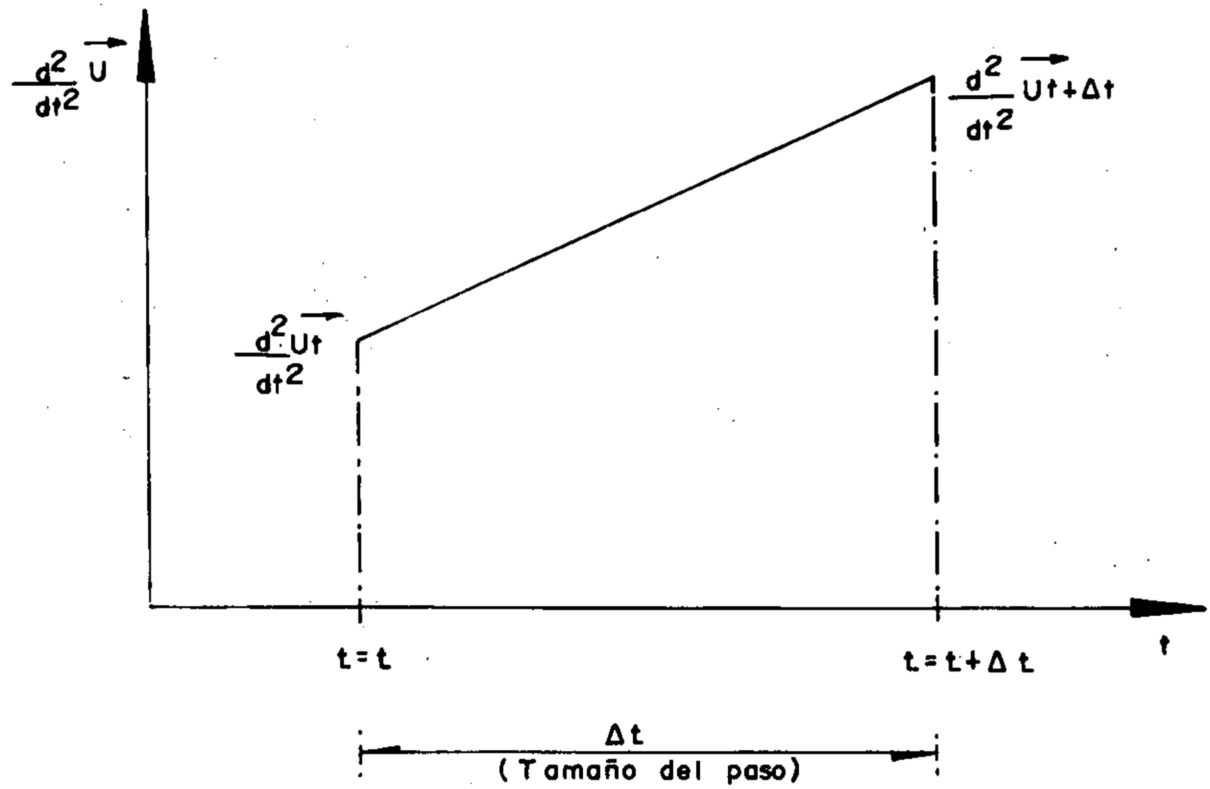


FIG 2.9 DIAFRAGMA RIGIDO UNIDO SOPORTADO LATERALMENTE POR LAS RIGIDECES DE ENTREPISO QUE LE SUBYACEN.



Valores conocidos al inicio del paso : $\vec{U}_t$, $\frac{d}{dt}\vec{U}_t$, $\frac{d^2}{dt^2}\vec{U}_t$

Valores por conocer al final del paso : $\vec{U}_{t+\Delta t}$, $\frac{d}{dt}\vec{U}_{t+\Delta t}$, $\frac{d^2}{dt^2}\vec{U}_{t+\Delta t}$

FIG. 4.1 Variación lineal de la aceleración en el intervalo de integración , Δt.

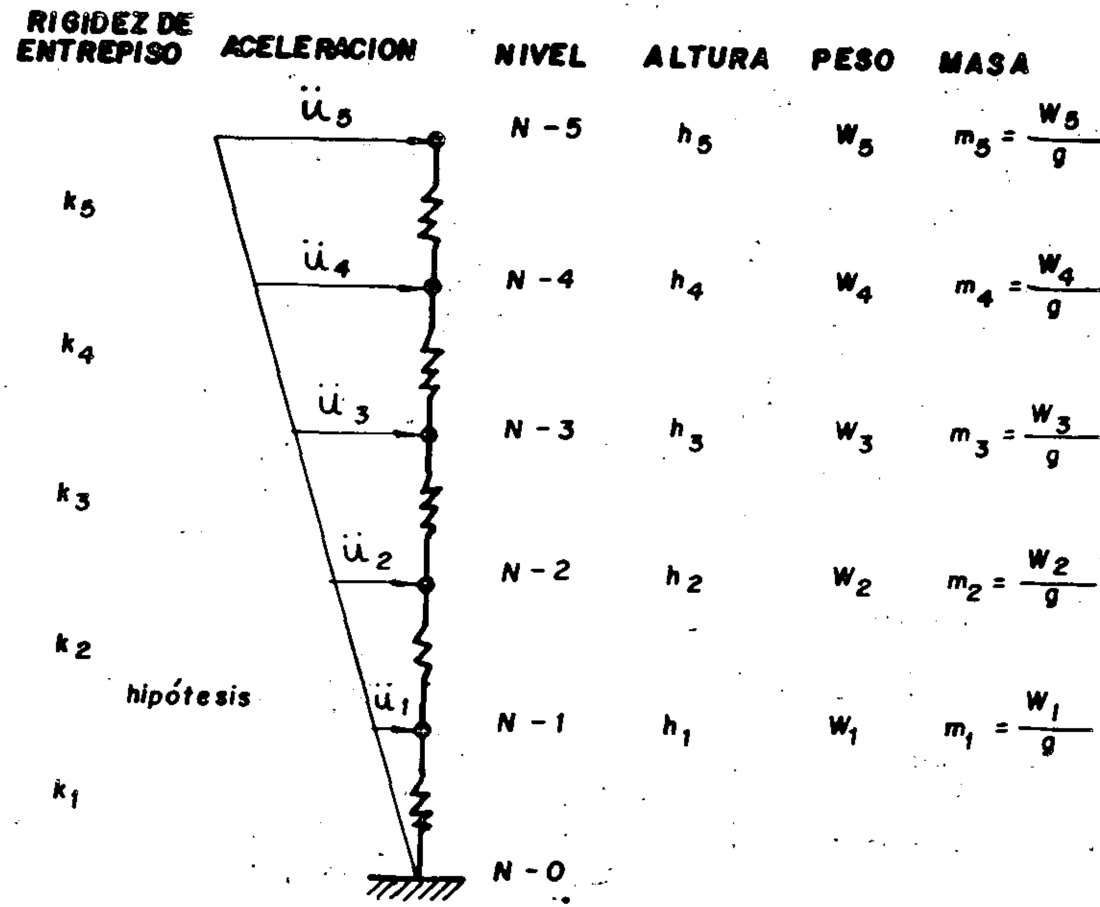


FIG 4.2 Distribución de Aceleraciones en el análisis estático.

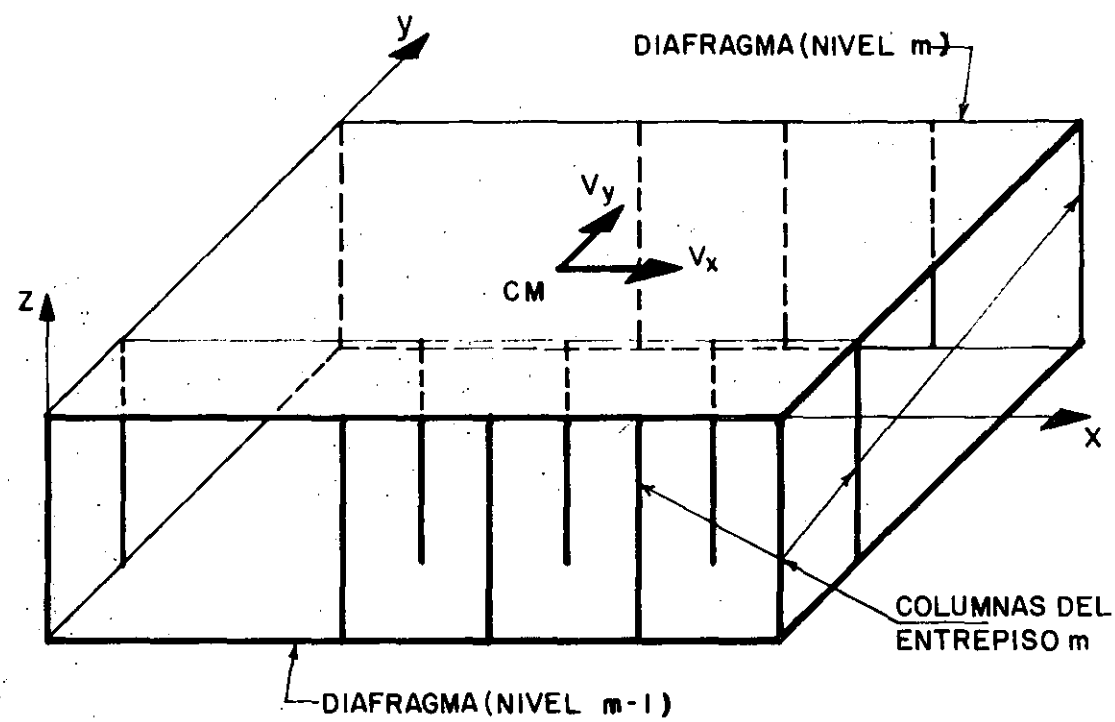


FIG. 5.1 REPRESENTACION ESQUEMATICA DEL MODELO ESTRUCTURAL CON RIGIDECES DE ENTREPISO.

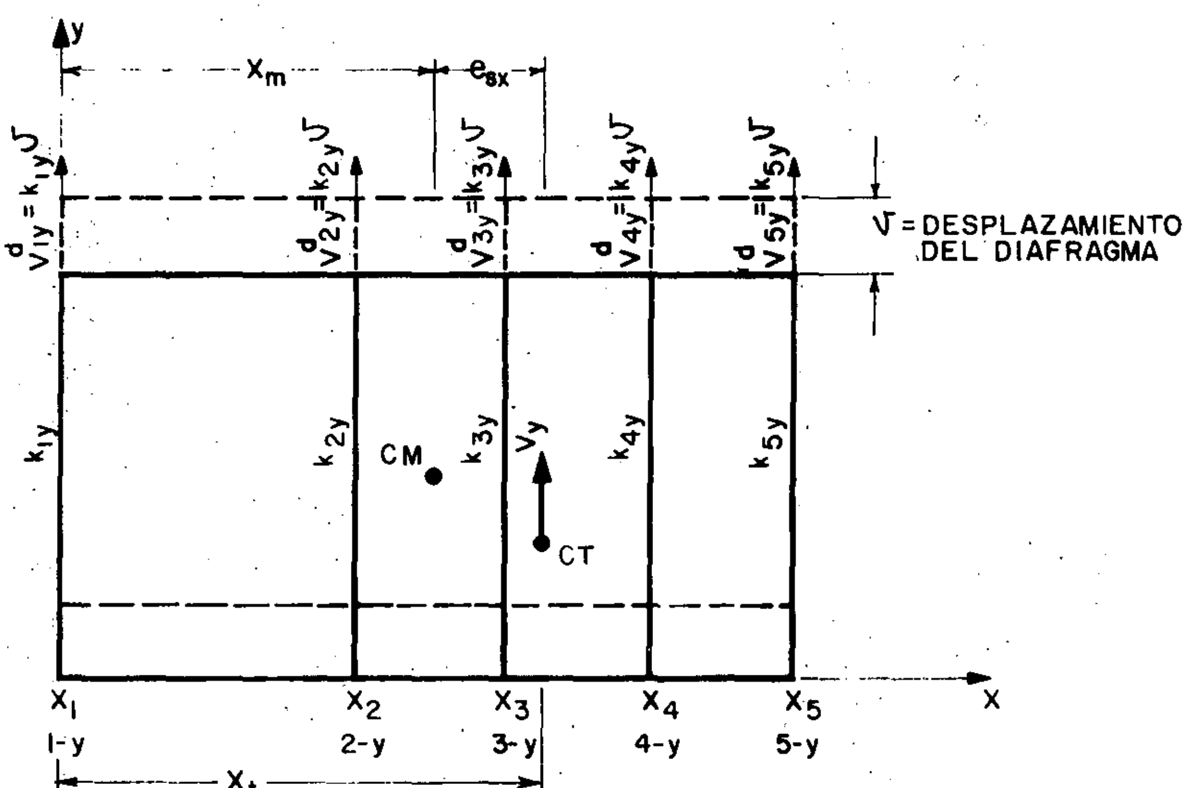


FIG. 5.2 FUERZAS CORTANTES DIRECTAS, V^{d}_{iy}, PARALELAS AL EJE y.

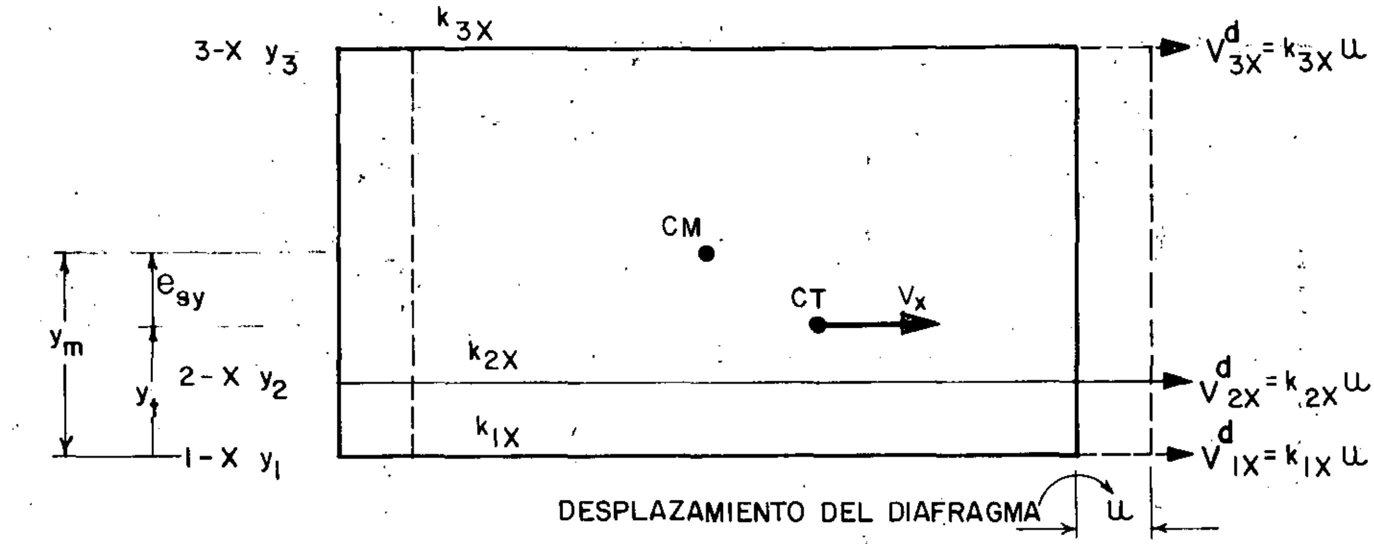


FIG. 5.3 FUERZAS CORTANTES DIRECTAS, V^{d}_{ix}, PARALELAS AL EJE X.

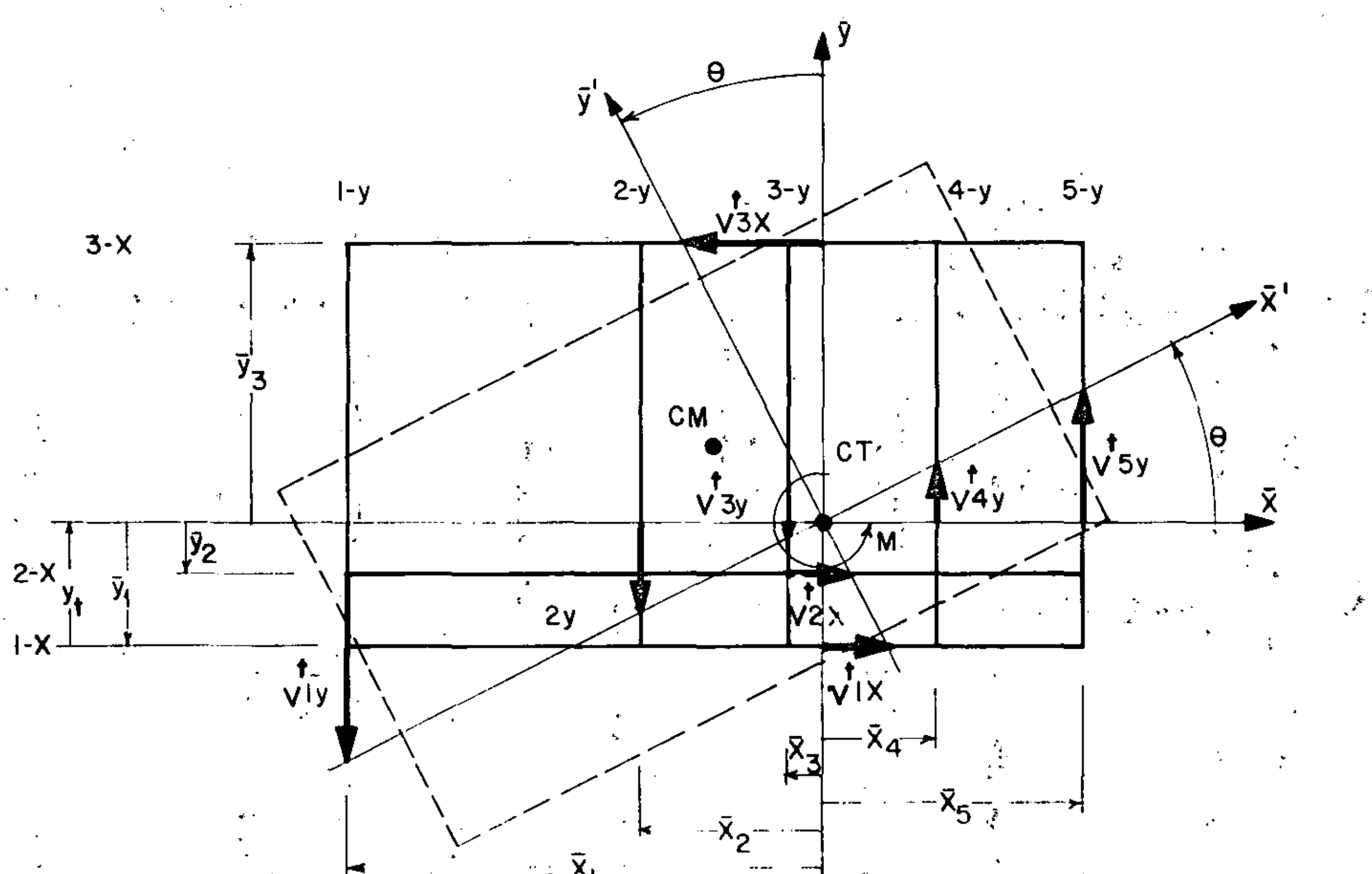
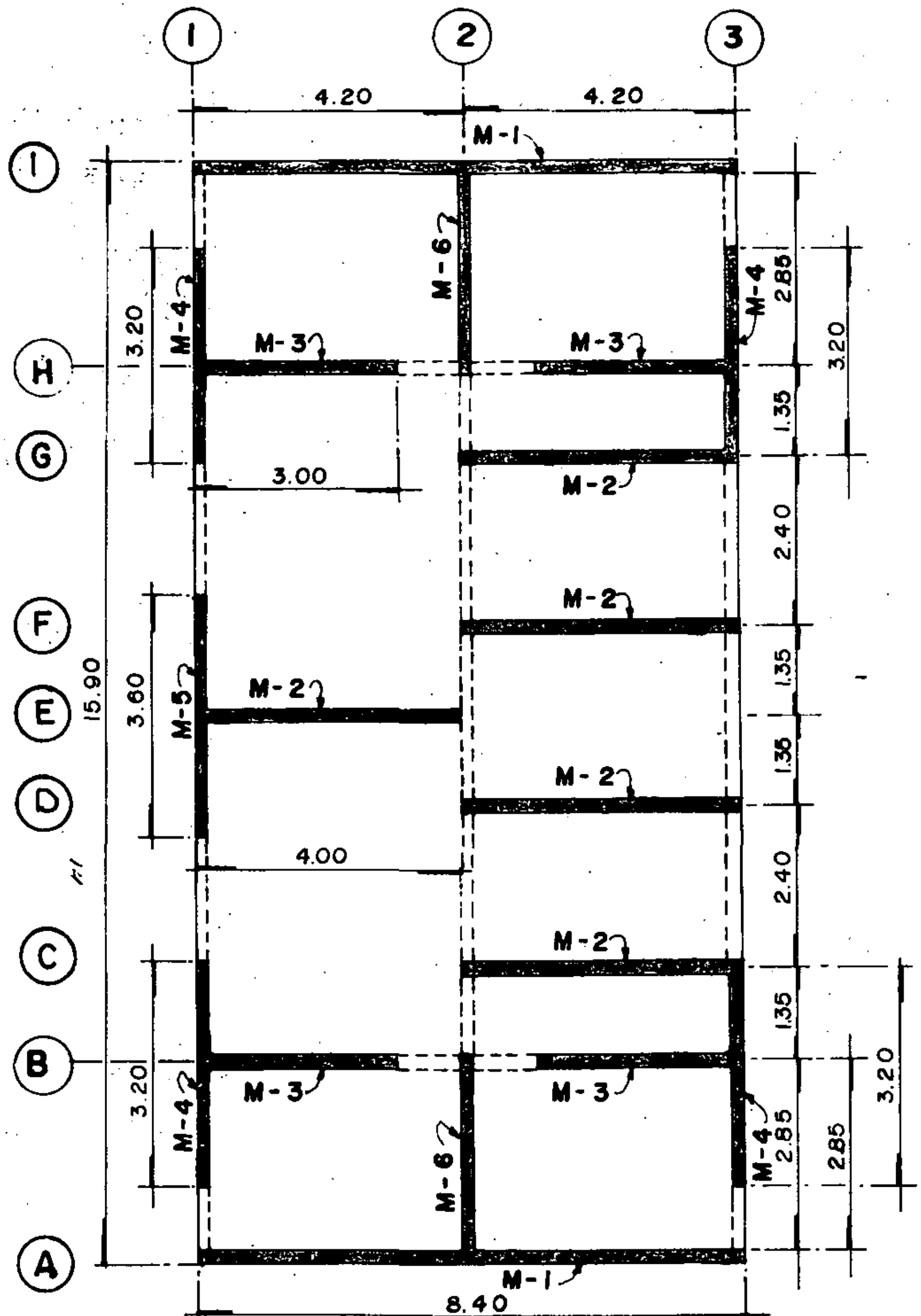
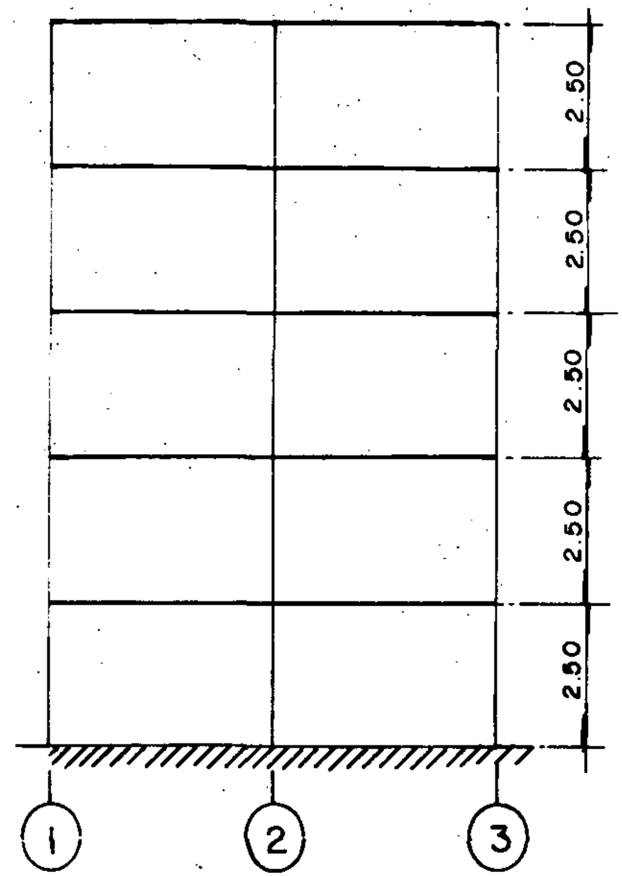


FIG. 5.4 FUERZAS CORTANTES DE TORSION, V^{t}_{ix} Y V^{t}_{jy}.

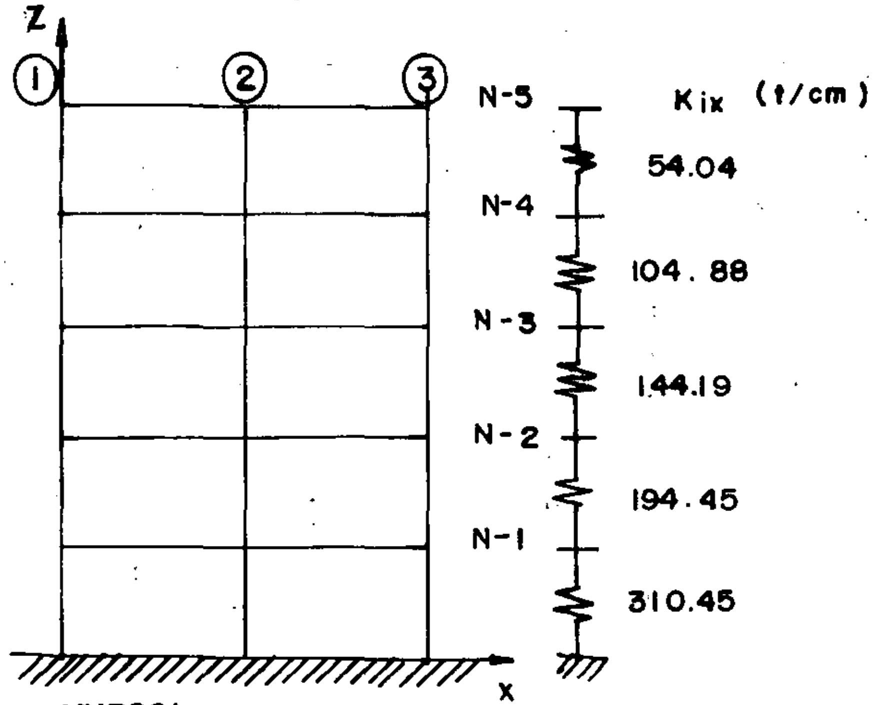


PLANTA TIPO ESC 1:100

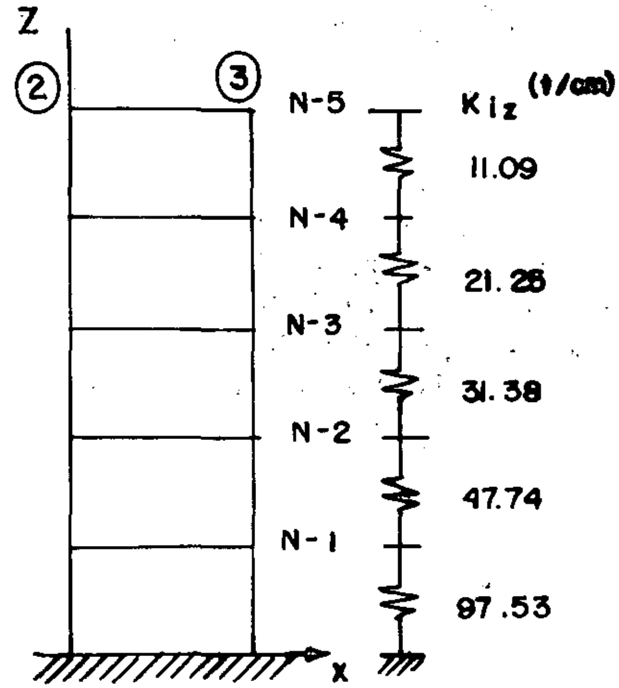


b) ELEVACION ESQUEMATICA DE NIVELES.

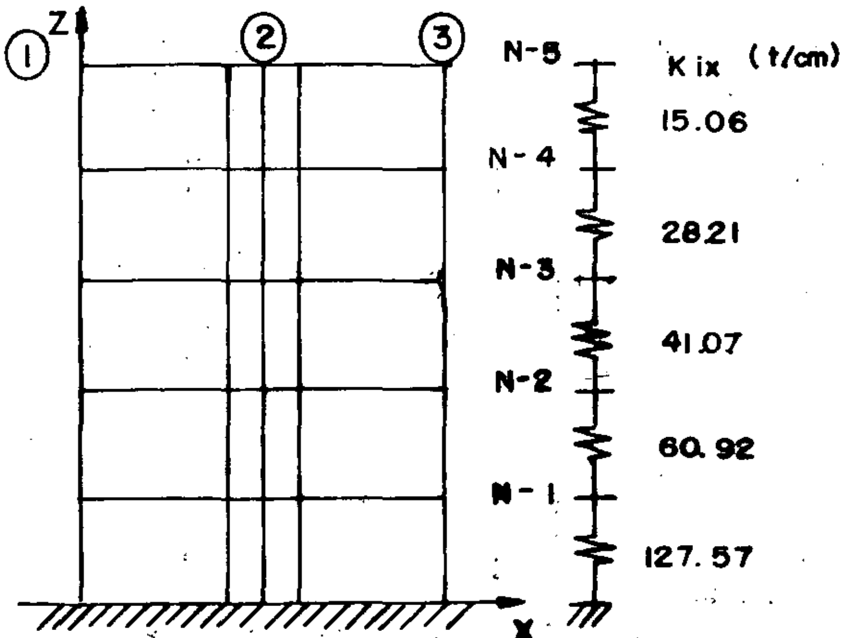
FIG 6.1 EDIFICIO PARA DESARROLLAR LOS EJEMPLOS.



MUROS: 1-X y 9-X

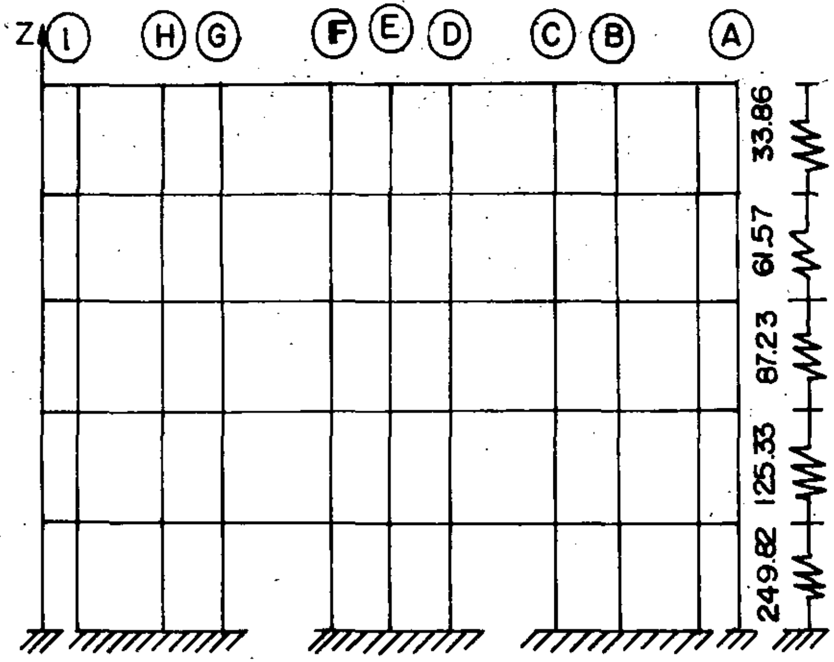


MUROS: 3-X, 4-X, 5-X, 6-X y 7-X.

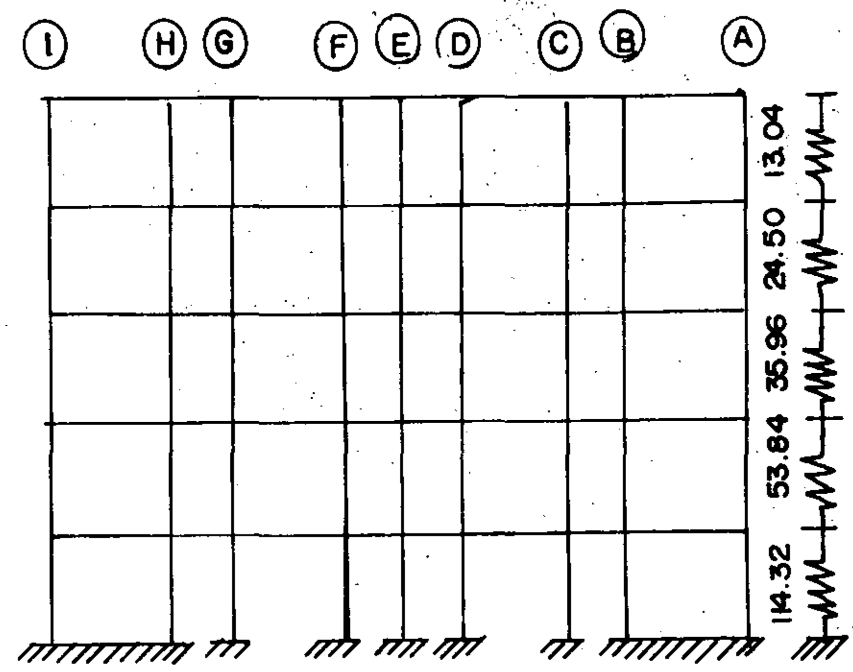


MUROS 2-X y 8X

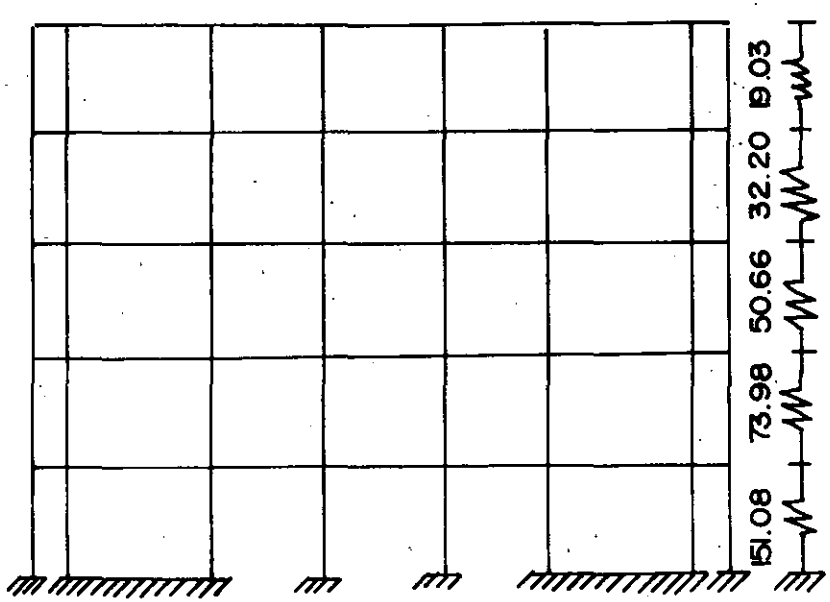
FIG 6.2 REPRESENTACION ESQUEMATICA MEDIANTE RIGIDECES DE ENTREPISO DE LOS MUROS PARALELOS AL PLANO X-Z.



MURO 1 – Y

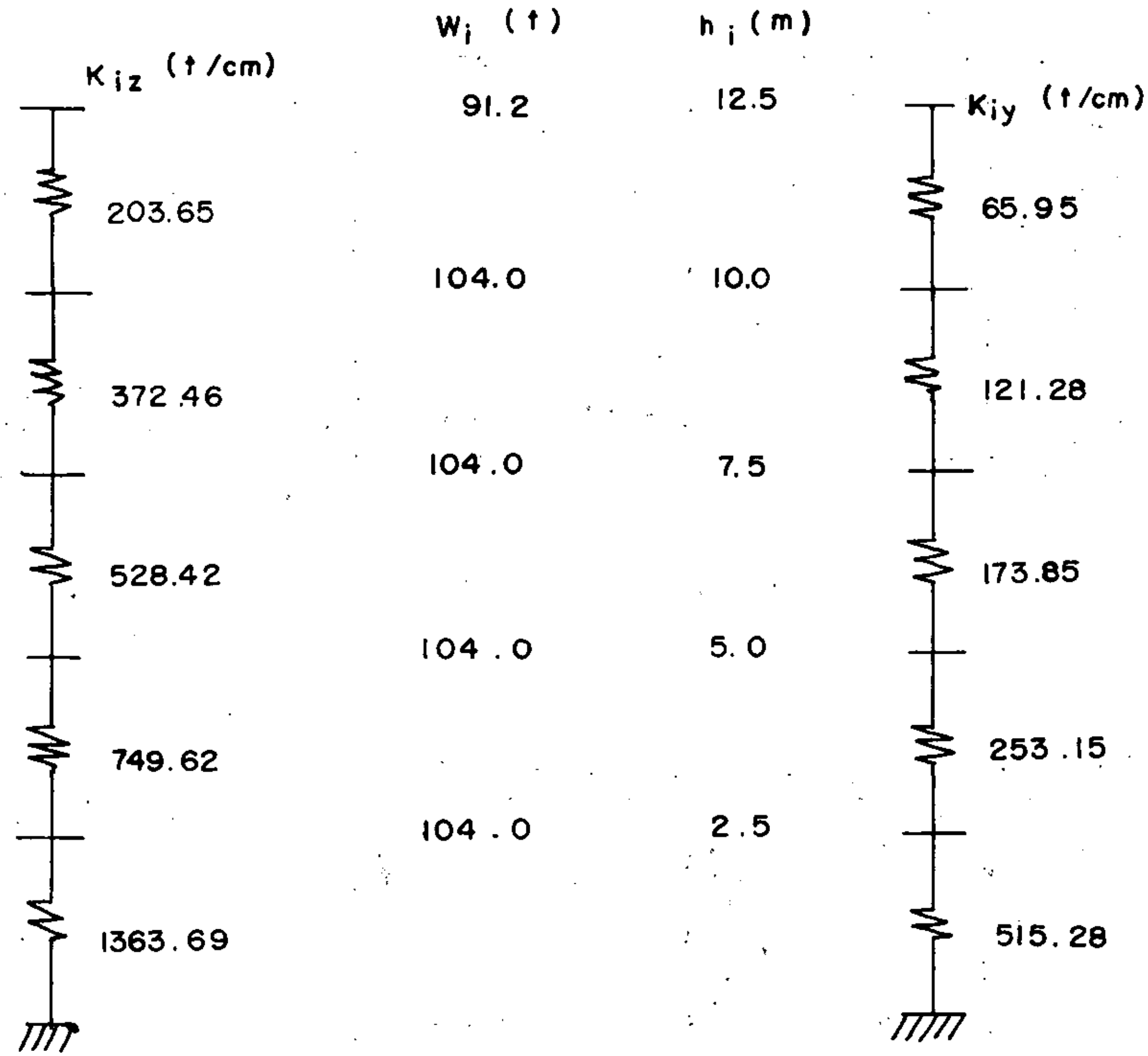


MURO 2 – y



MURO 3 – y

FIG 6.3 REPRESENTACION ESQUEMATICA MEDIANTE RIGIDECES DE ENTREPISO DE LOS MUROS PARALELOS AL PLANO y z.



a) PARALELO AL EJE X

b) PARALELO AL EJE y

FIG 6.4 MODELOS ESTRUCTURALES UNIDIMENSIONALES DEL EDIFICIO A BASE DE RIGIDECES DE ENTREPISO.

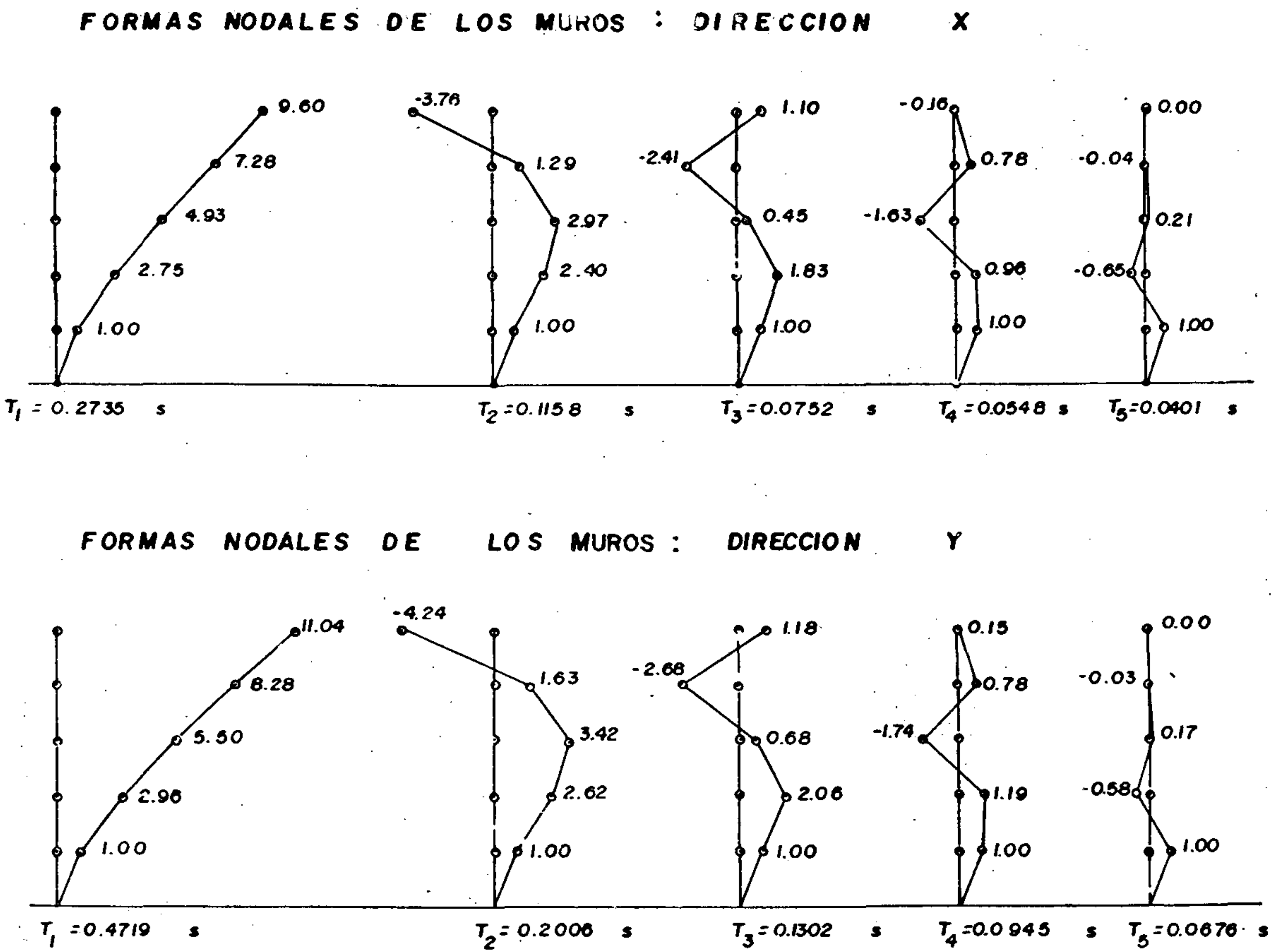


FIG 6.5 FORMAS MODALES (EIGENVECTORES) DE LOS MODELOS ESTRUCTURALES UNIDIMENSIONALES DEL EDIFICIO.

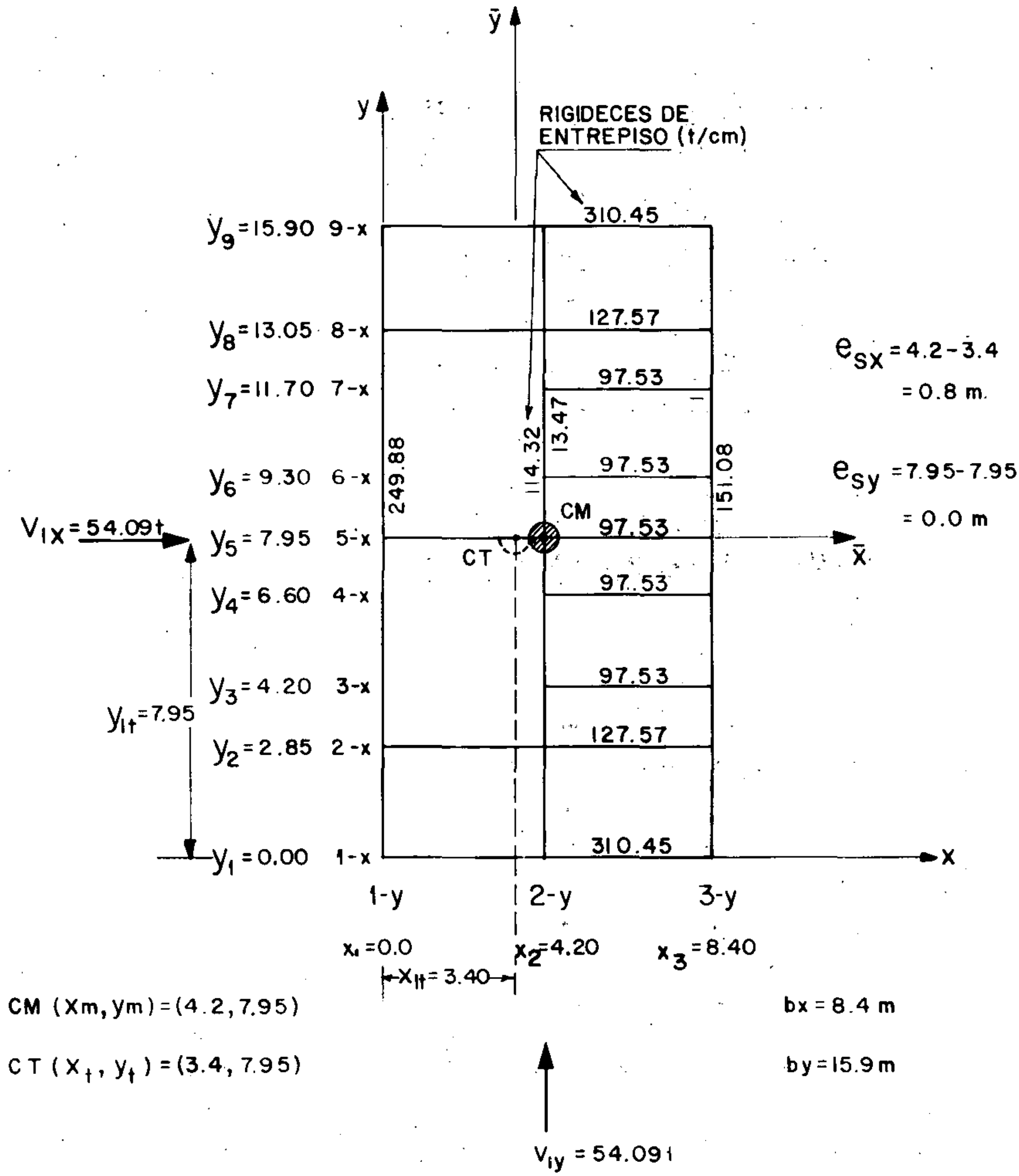
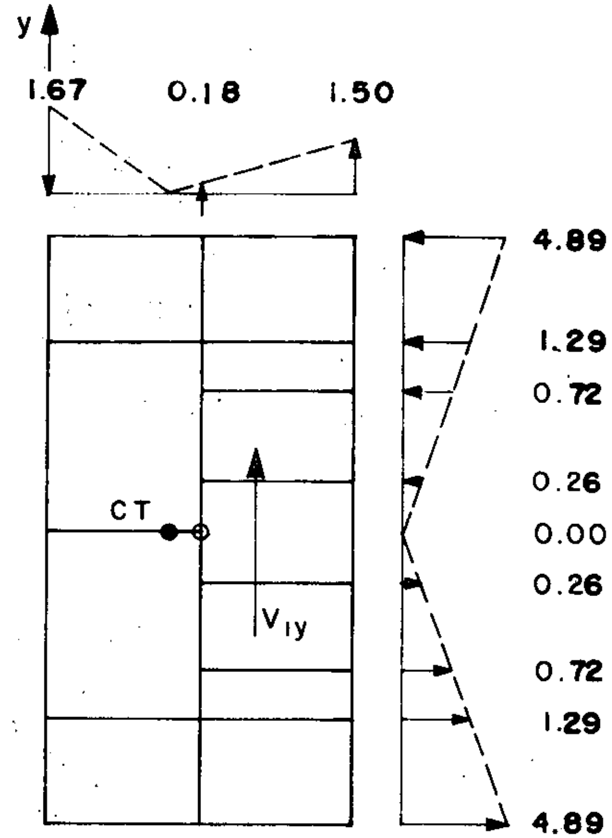
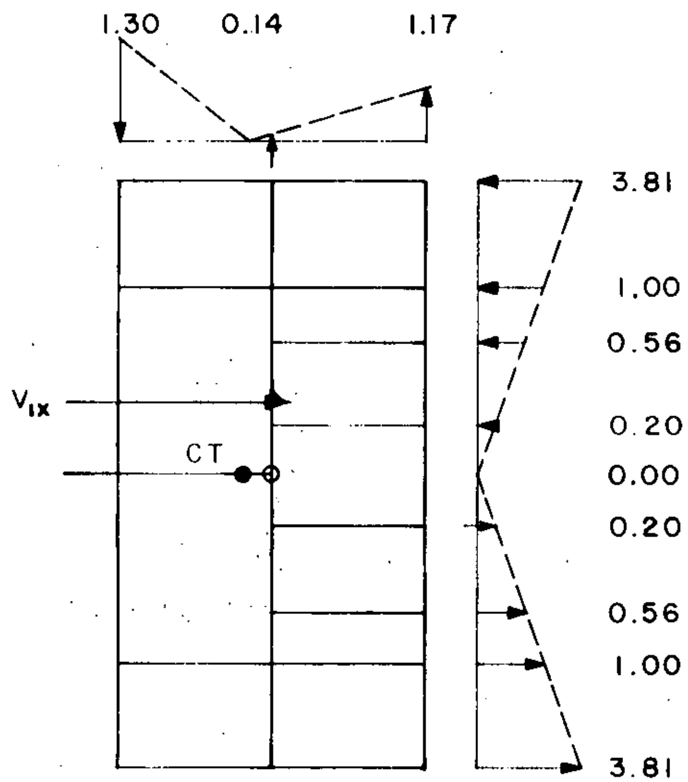


FIG 6.6 DISTRIBUCION DE LAS RIGIDECES DE ENTREPISO QUE LLEGAN AL NIVEL 1.



a) MOMENTO TORSIONANTE IGUAL A 110.34 tm.



b) MOMENTO TORSIONANTE IGUAL A 86.00 tm.

FIG 6.7 FUERZAS CORTANTES DEBIDAS A LA TORSION EN EL NIVEL 1.

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

CURSOS ABIERTOS

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

MODULO II:
ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMO

VIBRACIONES DE SISTEMAS DISCRETOS DE VARIOS GRADOS DE LIBERTAD

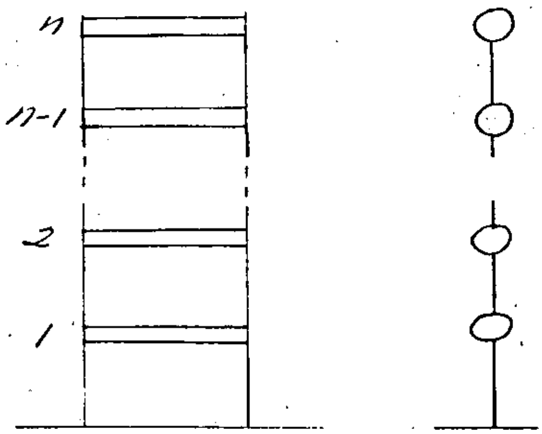
M. EN I. JOSE LUIS TRIGOS
M. EN I. JORGE PRINCE

Palacio de Minería Calle de Tacuba 5 Primer piso Deleg. Cuauhtémoc 06000 México, D.F. APDO. Postal M-2285
Teléfonos: 512-8955 512-5121 521-7335 521-1987 Fax 510-0573 521-4020 AL 26

VIBRACION DE SISTEMAS DISCRETOS DE VARIOS GRADOS DE LIBERTAD

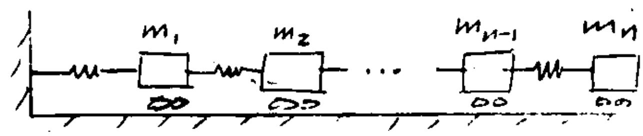
M. en I. Jorge Prince

Ejemplos de sistemas de n GL

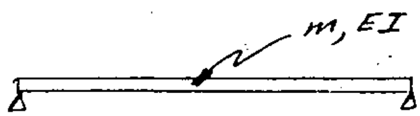


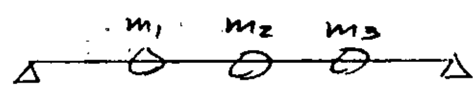
Características:

- masas $\begin{cases}\text{concentradas}\\ \text{rígidas}\\ \text{constantes}\end{cases}$
- columnas solo se deforman lateralmente
- con una coordenada por masa queda definida la configuración del sistema
- equivale a:

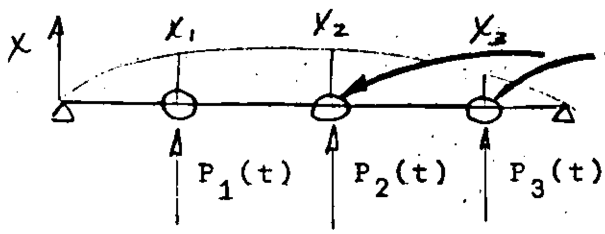


Además, la consideramos elástica, lineal





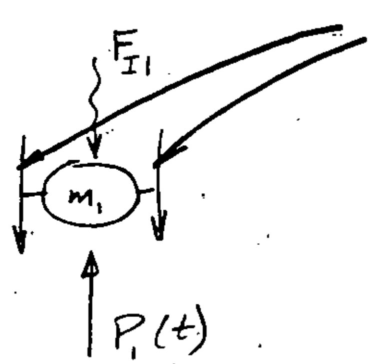
Supongamos:



masas concentradas

fuerzas concentradas exteriores

aislemos una masa:



$F_{r1} = \sum$ fuerzas resistencia elástica a la deformación

F_I = Fuerza de inercia

Las ecuaciones condensadas de movimiento serán:

$$F_{I1} + F_{r1} = P_1(t)$$

$$F_{I2} + F_{r2} = P_2(t)$$

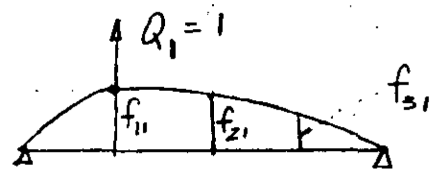
$$F_{I3} + F_{r3} = P_3(t)$$

Fuerzas asociadas al desplazamiento, <u>NO al movimiento</u>

∴ la determinación de estas fuerzas es un problema estático.

<u>Coeficientes de influencia</u>

1. De flexibilidad



f_{ij} = despl. de la coord. i debido a una carga unitaria en coord. j (desplazamiento y fuerza en = dirección)

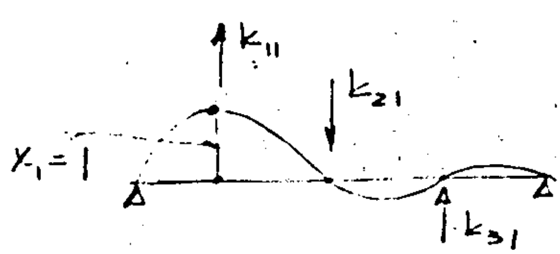
superposición

$$X_1 = f_{11}\,Q_1 + f_{12}\,Q_2 + f_{13}\,Q_3$$

$$X_2 = f_{21}\,Q_1 + f_{22}\,Q_2 + f_{23}\,Q_3 \qquad \text{inv. (1)}$$

$$X_3 = f_{31}\,Q_1 + f_{32}\,Q_2 + f_{33}\,Q_3$$

2. De rigidez:



K_{ij} = fuerza en coordenada i por un desplazamiento unitario en coordenada j.

Por superposición

$$Q_1 = K_{11}\ X_1 + K_{12}\ X_2 + K_{13}\ X_3$$

$$Q_2 = K_{21}\ X_1 + K_{22}\ X_2 + K_{23}\ X_3 \qquad (2)$$

$$Q_3 = K_{31}\ X_1 + K_{32}\ X_2 + K_{33}\ X_3$$

Desde luego $K_{ij} = K_{ji}$ (y $f_{ij} = f_{ji}$) (Maxwell-Mohr)

La ecuación 2 también puede escribirse:

$$Q_i = \sum_{j=1}^{3} K_{ij}\ X_j$$

o bien, en notación matricial

$$\begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \end{Bmatrix} \qquad (3)$$

matriz de rigideces

Ponemos:

$$\{Q\} = [K]\ \{X\}$$

Claro que $[K]^{-1} = [F] = [f_{ij}]$

Sustituyendo (2) o (3) en ecuaciones de movimiento:

$$m_1\ddot{X}_1 + K_{11}X_1 + K_{12}X_2 + K_{13}\ X_3 = P_1(t)$$

$$m_2\ddot{X}_2 + K_{21}X_1 + K_{22}X_2 + K_{23}\ X_3 = P_2(t)$$

$$m_3\ddot{X}_3 + K_{31}X_1 + K_{32}X_2 + K_{33}\ X_3 = P_3(t)$$

o bien:

$$\begin{bmatrix} m_1 & 0 & 0 \\ 0 & m_2 & 0 \\ 0 & 0 & m_3 \end{bmatrix} \begin{Bmatrix} \ddot{X}_1 \\ \ddot{X}_2 \\ \ddot{X}_3 \end{Bmatrix} + \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \end{Bmatrix} = \begin{Bmatrix} P_1(t) \\ P_2(t) \\ P_3(t) \end{Bmatrix}$$

o también:

$$[M] \{\ddot{X}\} + [K] \{X\} = \{P(t)\} \quad \text{(vibración forzada)}$$
$$= \{0\} \quad \text{(vibración libre)}$$

1. <u>VIBRACION LIBRE</u>

$$[M] \{\ddot{X}\} + [K] \{X\} = \{0\} \tag{1.1}$$

Supongamos la solución

$$\{X\} = \underbrace{\{r\}}_{\text{constante con } t} \underbrace{(A \text{ sen } pt + B \text{ sen } pt)}_{\text{escalar}} = \{r\} \; Y(t)$$

escalar → define:
- variación armónica
- amplitud

Obtenemos:

$$\{X\} = \{r\} \; (A \text{ sen } pt + B \cos pt) = r \quad Y(t)$$
$$\{\dot{X}\} = \{r\} \; (Ap \cos pt - B \, p \text{ sen } pt) \tag{1.2}$$
$$\{\ddot{X}\} = \{r\} \; (-Ap^2 \text{ sen } pt - B \, p^2 \cos pt) = -\, p^2 \{r\} \; Y(t)$$

Sustituyendo 1.2 en 1.1 y dividiendo entre Y(t) nos queda:

$$-\, p^2 [M] \{r\} + [K] \{r\} = \{0\}$$

o sea:

$$\Big[\underbrace{[K] - p^2 [M]}_{[E]} \Big] \{r\} = \{0\} \tag{1.3}$$

$[K]\{r\} = p^2 [M] \{r\}$

pre x $[M]^{-1}$

$[M]^{-1} [K]\{r\} = p^2 \{r\}$

$[K]\{r\} = p^2 [M] \{r\}$

pre x $[K]^{-1} \cdot \frac{1}{p^2}$

$\frac{1}{p^2} \{r\} = [K]^{-1} [M] \{r\}$

En las dos formas llegamos a un problema de VAC

$$[L] \{u\} = \lambda \{u\}$$

Problema de valores característicos:

- Dada una matriz cuadrada de orden (nxn) $[L]$, que representa una transformación lineal de vectores n-dimensionales, debe encontrarse un vector $\{u\}$ que transformado por $[L]$ resulte en otro vector $\lambda \{u\}$ en la misma "dirección". O sea, $[L]$ solo cambia la magnitud de $\{u\}$ sin cambiar la dirección.

  El vector es un vector característico (o eigenvector) de $[L]$.
  λ(escalar) representa la relación entre las "longitudes" antes y después de la transformación y para llegar a los VEC debe tomar valores de un conjunto de valores característicos (VAC) (o eigenvalores).

  El problema de encontrar frecuencias y modos naturales puede considerarse un problema de VAC. - (STD)

Tenemos

$$\left[[K] - p^2 [M]\right] \{r\} = \{0\} \tag{1.3}$$

Si en el sistema de ecuaciones

$$[A] \{x\} = \{0\}$$

$[A]$ es no singular, la solución única es la trivial $\{X\} = \{0\}$, de donde nos interesa el caso en que $[A]$ es singular. En este caso la adjunta* $[\hat{A}]$ existe y puede pre X por ella, con el resultado

$$|A| \{X\} = \{0\}$$

porque $[\hat{A}] \; [A] = |A| \; [I] \quad \forall [A] \;$ (nxn)

Puesto que $|A| = 0$, $\{x\}$ no necesariamente es nulo, pero si se asigna un valor dado a uno de sus elementos los demás quedan determinados en forma única.

También notamos que si $\{X\}$ es solución de $[A] \{X\} = \{0\}$ y α es una constante, entonces $\alpha\{X\}$ es también solución. Por lo tanto, hay un número infinito de soluciones. Todos estas se considerarán juntas y hablaremos de una "solución" como un conjunto de relaciones entre los elementos de $\{x\}$.

Volvemos a $\Big[\underbrace{[K] - p^2 [M]}_{[E]}\Big] \{r\} = \{0\}$ (1.3)

Al desarrollar $|E| = 0$ llegamos a una ecuación de grado n en p^2, cuyas raíces son los VAC.

- Como $[K]$ y $[M]$ son simétricas y positivas definidas*,

---

*Transpuesta de la matriz de cofactores.

**$[A]$ es POS. DEF. si $\{\dot{q}\} \; [A] \; \{\dot{q}\} > 0$ para todo $\{\dot{q}\}$ no nulo puede demostrarse que las raîces de la ecuación característica son reales y positivas. Las llamamos p_1^2, p_2^2, ..., p_n^2.

Las n frecuencias naturales son los términos positivos de las raîces y la más baja es llamada <u>frecuencia fundamental</u>.

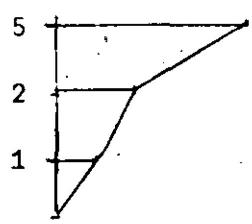
- Para la gran mayoría de los casos de interés las frecuencias son diferentes entre sí.
- Para cada frecuencia p_i existe una VEC asociado:

$$[K]\ \{r\}_i = p_i^2\ [M]\ \{r\}_i \qquad i = 1, \ldots, n$$

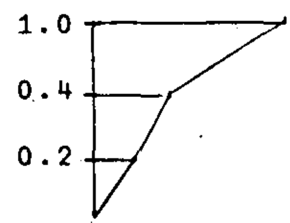
o sea para cada p_i existe una solución $\{r\}$ no trivial

- Normalización (solo conveniencia, sin significado físico)

Varias formas:



$$\{r\}_i^t\ [M]\ \{r\}_i = 1$$



(modos normales)

- Los modos y frecuencias naturales del sistema son propiedades características derivadas de las propiedades de inercia y rigidez expresadas por los elementos de $[M]$ y $[K]$.
- Llamaremos <u>matriz modal</u> $[R]$ a la que tiene los VEC, o vectores modales, como columnas.

ORTOGONALIDAD DE MODOS DE VIBRACION

Se dice que dos vectores $\{a\}$ y $\{b\}$ son <u>ortogonales</u> con respecto a la matriz simétrica $[J]$ si

$$\{a\}'\ [J]\ \{b\} = \{b\}'\ [J]\ \{a\} = 0$$

Demostremos que dos vectores modales $\{r\}_i$ y $\{r\}_j$, asociados a frecuencias diferentes ($p_i \neq p_j$) son ortogonales con respecto a las matrices de inercia y elástica.

- Cada uno de estos vectores satisface la ecuación 1.3

$$p^2\ [M]\{r\} = [K]\{r\} \qquad [M]\ \{r\} = \frac{1}{p^2}[K]\ \{r\}$$

es decir:

$$p_i^2\ [M]\ \{r\}_i = [K]\ \{r\}_i \qquad [M]\ \{r\}_i = \frac{1}{p_i^2}[K]\ \{r\}_i$$

$$p_j^2\ [M]\ \{r\}_j = [K]\ \{r\}_j \qquad [M]\ \{r\}_j = \frac{1}{p_j^2}[K]\ \{r\}_j$$

[illegible] i y j por $\{r\}'_j$ y $\{r\}'_i$ respectivamente

$$\left.\begin{array}{l} p_i^2\ \{r\}'_j\ [M]\ \{r\}_i = \{r\}'_j [K]\{r\}_i \\ p_j^2\ \{r\}'_i\ [M]\ \{r\}_j = \{r\}'_i [K]\{r\}_j \end{array}\right\} \begin{array}{l} \{r\}'_j [M]\{r\}_i = \dfrac{1}{p_i^2}\ \{r\}'_j [K]\{r\}_i \\ \{r\}'_i [M]\{r\}_j \quad \dfrac{1}{p_j^2}\ \{r\}'_i [K]\{r\}_j \end{array} \qquad (a)$$

pero como $[M]$ y $[K]$ son simétricas:

$$\{r\}'_j\ [K]\ \{r\}_i = \{r\}'_i\ [K]\{r\}_j$$
$$\{r\}'_j\ [M]\ \{r\}_i = \{r\}'_i\ [M]\{r\}_j$$

$\therefore$, restando miembro a miembro en ecuaciones (a):

$$(p_i^2 - p_j^2)\left(\{r\}_i' \; [M] \; \{r\}_j\right) = 0 \qquad 0 = \left(\frac{1}{p_i^{\,2}} - \frac{1}{p_j^{\,2}}\right)\{r\}_i' [K] \{r\}_j$$

y como $p_i^{\,2} \neq p_j^{\,2}$

$$\{r\}_i' \; [M] \; \{r\}_j = 0 \qquad \{r\}_i' \; [K] \; \{r\}_j = 0$$

Tenemos ecuaciones de ortogonalidad:

$$\left.\begin{aligned} \{r\}_i' \; [M] \; \{r\}_j &= 0 \\ \{r\}_i' \; [K] \; \{r\}_j &= 0 \end{aligned}\right\} \quad \text{si } i \neq j$$

La ec

$$[M] \; \{\ddot{x}\} + [K] \{x\} = \{0\} \tag{a}$$

y la matriz modal $[R]$

Hagamos:

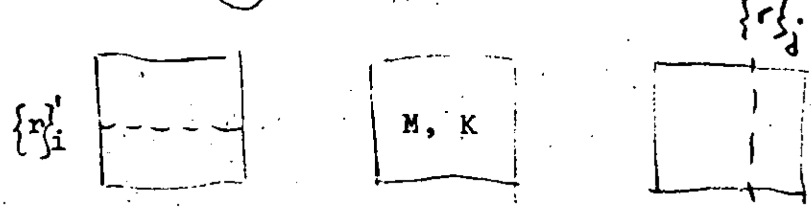
$$\{x\} = [R] \; \{y\}$$

y sustituyendo en (a):

$$[M] \; [R] \; \{\ddot{y}\} + [K] \; [R] \; \{y\} = \{0\}$$

premultiplicando por $[R]'$:

$$\underbrace{[R]'[M] \; [R]}_{\text{diagonales}} \{\ddot{y}\} + \underbrace{[R]' \; [K] \; [R]}_{\text{diagonales}} \{y\} = \{0\} \tag{b}$$



$i \neq j$

Llamemos

$$[R]'\,[M]\,[R] = [M^*]$$

$$[R]'\,[K]\,[R] = [K^*]$$

$\therefore$ la ec (b) (p. 14) puede ponerse:

$$[M^*]\,\{\ddot{y}\} + [K^*]\,\{y\} = \{0\}$$

que equivale a:

$$m_{11}^{*}\,\ddot{y}_1 + k_{11}^{*}\,y_1 = 0$$

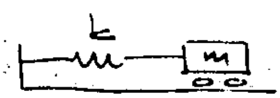
$$m_{22}^{**}\,\ddot{y}_2 + k_{22}^{*}\,y_2 = 0$$

- - - - - - - - - - - -

$$m_{nn}^{*}\,\ddot{y}_n + k_{nn}\,y_n = 0$$

de las que

$$p_1^{\,2} = \frac{k_{11}^{*}}{m_n^{*}}\,, \ldots\ldots, \quad p_n^{\,2} = \frac{k_{nn}^{*}}{m_{nn}^{*}}$$

Recordar que para $\longrightarrow$ 

$$m\ddot{x} + kx = 0$$

$$\ddot{x} + p^2 x = 0 \quad y \quad p^2 = \frac{k}{m}$$

O sea, con la transformación

$$\{x\} = [R]\,\{y\}$$

aplicada a la ecuación

$$[M]\,\{\ddot{x}\} + [K]\,\{x\} = \{0\}$$

hemos descompuesto un sistema de nGL en <u>n sistemas de 1GL independientes</u>.

Consideremos el producto

$$[M^*]^{-1}\,[K^*] = \left([R]'\,[M]\,[R]\right)^{-1}\,[R]'\,[K]\,[R] = [K^*]\,[M^*]^{-1}$$

$$= [R]^{-1}\,[M]^{-1}\,[R']^{-1}\,[R]'\,[K]\,[R]$$

$$= [R]^{-1}\,[M]^{-1}\,[K]\,[R] = \lceil \phi \rfloor$$

$\lceil \phi \rfloor$ contiene las frecuencias naturales en la diagonal principal

$\therefore$ El problema de encontrar frecuencias y modos naturales equivale al de encontrar la matriz $[R]$ que diagonalice $[M]$ y $[K]$ de acuerdo con

$$[R]'\,[M]\,[R] = \lceil M^* \rfloor$$
$$[R]'\,[K]\,[R] = \lceil K^* \rfloor$$

Las frecuencias naturales se obtendrán de

$$\lceil M^* \rfloor^{-1}\,\lceil K^* \rfloor = \lceil K^* \rfloor\,\lceil M^* \rfloor^{-1} = \lceil \phi \rfloor$$

Veámoslo en otra forma

$$\lceil M \rfloor\,\{\ddot{x}\} + [K]\,\{x\} = \{P(t)\}$$

Sustituyendo $\{x\} = [R]\,\{y\}$

$$[M]\,[R]\,\{\ddot{y}\} + [K]\,[R]\,\{y\} = \{P(t)\}$$

premultiplicando por $\{r\}'_j$

$$\underbrace{\{r\}'_j \; [M] \; [R]}_{(a)} \; \{\ddot{y}\} + \underbrace{\{r\}'_j \; [K] \; [R]}_{(b)} \; \{y\} = \underbrace{\{r\}'_j \; \{P(t)\}}_{\text{escalar}}$$

En los productos (a) y (b) solo queda (por ortogonalidad):

$$\underbrace{\{r\}'_j \; [M] \; \{r\}_j}_{M^*_j} \; \ddot{y}_j + \underbrace{\{r\}'_j \; [K] \{r\}_j}_{K^*_j \,=\, p^2_j \; M^*_j} \; y_j = \underbrace{\{r\}'_j \; \{P(t)\}}_{P^*_j \,=\, \sum_i P_i \, r_{ij}}$$

y para el modo j tenemos:

$$M^*_j \; \ddot{y}_j + p_j^2 \; M^*_j \; y_j = P^*_j(t)$$

o bien (1.5)

$$M^*_j \; \ddot{y}_j + K^*_j \; y_j = P^*_j(t)$$

análoga a la ecuación de movimiento para 1 GL:

$$m \ddot{x} + k x = P(t)$$

En (1.5) tenemos:

n ecuaciones independientes para nGL

1 ecuación independiente para cada modo

Para vibración libre (1GL)

$$\ddot{x} + p^2 x = 0 \qquad p^2 = \frac{k}{m}$$

la solución es:

$$x = A \cos pt + B \text{ sen } pt \qquad (c)$$

y para el modo j tendremos $(P_j(t) = 0)$

$$y_j = A_j \cos p_j t + B_j \text{ sen } P_j t \qquad (d)$$

Si en (c) hacemos

$$x\big]_{t=0} = X_o \qquad \dot{x}\big]_{t=0} = \dot{X}_o$$

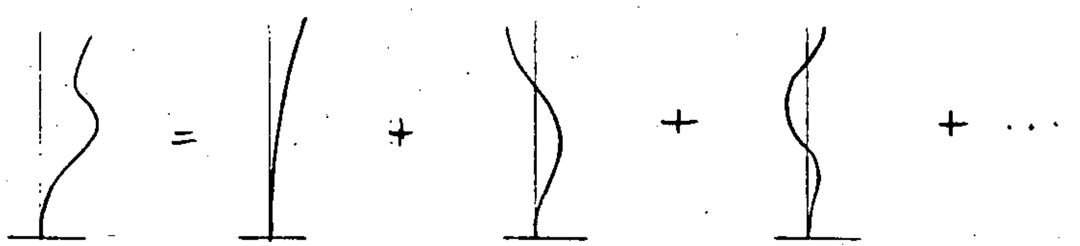
llegamos a

$$x(t) = X_o \cos pt + \frac{\dot{X}_o}{p} \text{ sen } pt$$

y $\therefore$ en (d):

$$y_j = y_{oj} \cos p_j t + \frac{\dot{y}_{oj}}{p_j} \text{ sen } p_j t$$

Cualquier configuración del sistema puede expresarse como una suma de formas modales multiplicadas por ciertos coeficientes. Esquemáticamente:



$$\{X\} = \{r\}_1 Y_1 + \{r\}_2 Y_2 + \{r\}_3 Y_3 + \ldots$$

estática
o
dinámica

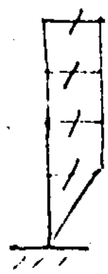
$$(Y = Y(t))$$

$$\left(\{X\} = \{X(t)\}\right)$$

En nuestra expresión

$$\{x\} = [R]\ \{y\} \qquad 1.4$$

$\{x\}$ puede no ser función de t, por ejemplo:



$$\{1\} = [R]\ \{c\} \qquad (e)$$

donde $\{c\}$ es el vector de constantes que prex $[R]$ nos da la configuración $\{1\}$

De la ec. (e):

$$\{c\} = [R]^{-1}\ \{1\} \qquad \left([R]\ \text{NO SING}\right)$$

En 1.4 también podríamos hacer

$$\{Y\} = [R]^{-1}\ \{x\}$$

pero sigamos otro camino, premultiplicando por $\{r\}'_j\ [M]$ y por $\{r\}'_j\ [K]$

$$\{r\}'_j\ [M]\ \{x\} = \{r\}'_j\ [M][R]\ \{y\} = \{r\}'_j\ [M]\ \{r\}_1\ y_1 +$$

$$+ \{r\}'_j\ [M]\ \{r\}_2\ y_2 + \cdots$$

$$\cdots + \{r\}'_j\ [M]\ \{r\}_n\ Y_n$$

Por ortogonalidad todos estos productos son nulos excepto el término

$$\{r\}'_j\ [M]\ \{r\}_j\ Y_j$$

de donde tenemos

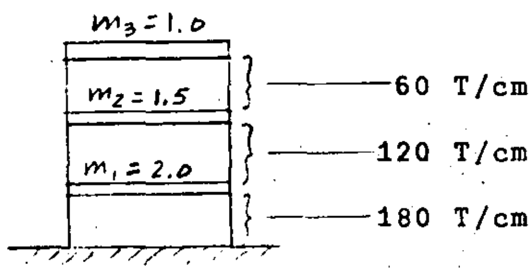
$$\{r\}_j' \cdot [M] \{x\} = \{r\}_j' [M] \{r\}_j \; y_j$$

de donde:

$$y_j = \frac{\{r\}_j' [M] \{x\}}{\{r\}_j' [M] \{r\}_j} = \frac{\{r\}_j [M] \{X\}}{M_j^*} = \frac{\{r\}_j' [K] \{x\}}{K_j^*} = \frac{\{r\}_j [K] \{x\}}{p_j^2 \; M_j^*}$$

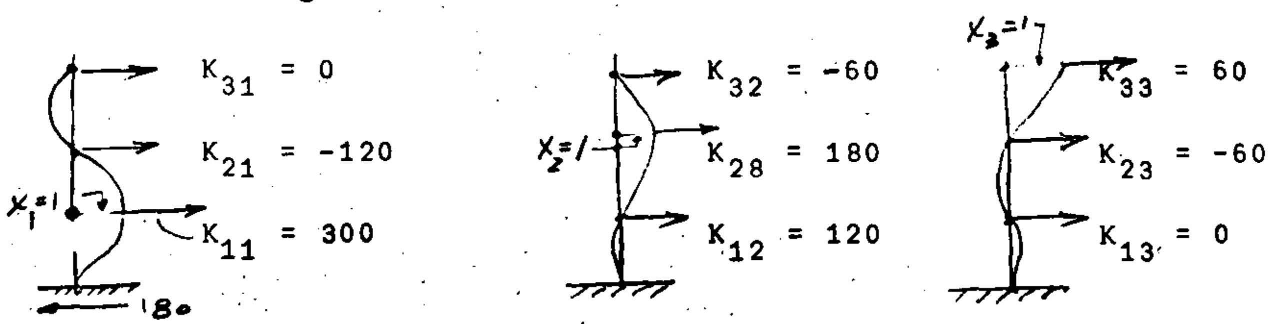
(coeficiente de participación)

Ejemplo (vigas rígidas)



$$[M] = \begin{bmatrix} 2.0 & 0 & 0 \\ 0 & 1.5 & 0 \\ 0 & 0 & 1.0 \end{bmatrix} \frac{\text{ton seg}^2}{\text{cm}}$$

Matriz de rigideces



$$\therefore [K] = \begin{bmatrix} 300 & -120 & 0 \\ -120 & 180 & -60 \\ 0 & -60 & 60 \end{bmatrix} = 60 \begin{bmatrix} 5 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix} \quad (\text{T/cm})$$

$$[E] = \left[[K] - p^2 \; [M] \right] \qquad M = \begin{bmatrix} 2 & 0 & 0 \\ 0 & 1.5 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$$= 60 \begin{bmatrix} (5 - \frac{2}{60} p^2) & -2 & 0 \\ -2 & (3 - \frac{1.5}{60} p^2) & -1 \\ 0 & -1 & (1 - \frac{1}{60} p^2) \end{bmatrix}$$

si $d = p^2/60$:

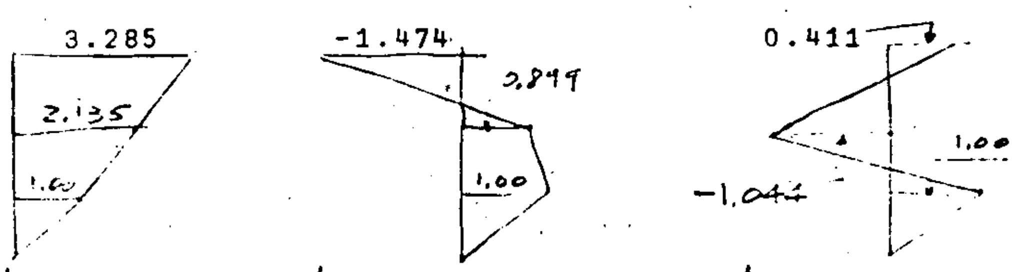
$$[E] = 60 \begin{bmatrix} (5-2d) & -2 & 0 \\ -2 & (3-1.5\,d) & -1 \\ 0 & -1 & (1-d) \end{bmatrix}$$

$$|E| = 0 = 60\,(d^3 - 5.5\,d^2 + 7.5\,d - 2) = 0$$

$$d_1 = 0.35$$
$$d_2 = 1.61$$
$$d_3 = 3.54$$

$$p^2 = 60\,d: \qquad \left. \begin{array}{ll} p_1^2 = 21.0 & p_1 = 4.58 \\ p_2^2 = 96.5 & p_3 = 9.82 \\ p_3^2 = 212.4 & p_3 = 14.56 \end{array} \right\} \text{frecuencias naturales}$$

Modos:



$p_1 = 4.58$ $\qquad$ $p_2 = 9.82$ $\qquad$ $p_3 = 14.56$

$$[R] = \begin{bmatrix} 1.000 & 1.000 & 1.000 \\ 2.135 & 0.899 & -1.044 \\ 3.285 & -1.474 & 0.411 \end{bmatrix}$$

$$[M^*] = [R]' \, [M] \, [R] = \begin{bmatrix} 19.629 & 0.038 & 0.007 \\ 0.037 & 5.386 & -0.014 \\ 0.006 & -0.014 & 3.804 \end{bmatrix}$$

Ej:

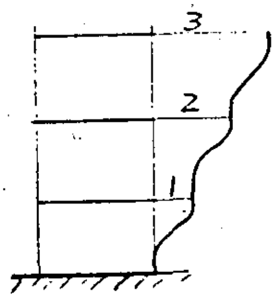
$$19.6296 = \{r\}_1' \, [M] \, \{r\}_1 = M_1^* = \sum_i r_{i1}^2 \, m_i$$

$$[K^*] = [R]' \, [K] \, [R] = 60 \begin{bmatrix} 6.899 & 0.042 & 0.034 \\ 0.042 & 8.651 & -0.040 \\ 0.034 & -0.040 & 13.473 \end{bmatrix}$$

Comprobación con $[K^*] = [p^2 \, M^*] =$

$$= \begin{bmatrix} 412.209 & 0 & 0 \\ 0 & 519.749 & 0 \\ 0 & 0 & 807.970 \end{bmatrix} = [p^2 \, M^*]$$

$$[K^*] = \begin{bmatrix} 413.940 & 0\cdots & 0\cdots \\ 0\cdots & 519.060 & 0\cdots \\ 0\cdots & 0\cdots & 808.380 \end{bmatrix}$$



$X_{30} = 3$ cm

$X_{20} = 2$ cm

$X_{10} = 1$ cm

$$\{X_o\} = \begin{Bmatrix} 1 \\ 2 \\ 3 \end{Bmatrix}$$

$$Y_{01} = \frac{\{r\}_1^t [M] \{x_o\}}{M_1^*} = \frac{2.0 + 6.405 + 9.855}{19.629} = 0.9303 \text{ cm}$$

$$Y_{02} = \frac{\{r\}_2 [M] \{x_o\}}{M_2^*} = \frac{2.0 + 2.697 - 4.422}{5.386} = 0.0511$$

$$Y_{03} = \frac{\{r\}_3 [M] \{x_o\}}{M_3^*} = \frac{2.0 - 3.132 + 1.233}{3.804} = 0.0266$$

Modo $Y_1(t)$

$P_1 = 4.58$

$P_2 = 9.82$

$P_3 = 14.56$

rn p.

0.930 cm
0.051 cm
0.026 cm

son amplitudes de los modos

Para obtener los desplazamientos de las masas debemos multiplicar por las configuraciones modales:

$$x_{i1} = \{r\}_1 \quad Y_1(t) = \begin{Bmatrix} 1.0 \\ 2.135 \\ 3.285 \end{Bmatrix} \quad 0.93 \cos 4.58\, t$$

$$x_{i2} = \{r\}_2 \quad Y_2(t) = \begin{Bmatrix} 1.0 \\ 0.899 \\ -1.474 \end{Bmatrix} \quad 0.051 \cos 9.82\, t$$

$$x_{i3} = \{r\}_3 \quad Y_3(t) = \begin{Bmatrix} 1.00 \\ -1.044 \\ 0.411 \end{Bmatrix} \quad 0.0266 \cos 14.56\, t$$

y sumar. O sea los desplazamientos $x_i(t)$ de las masas serán

$$\{x(t)\} = [R]\{y(t)\}$$

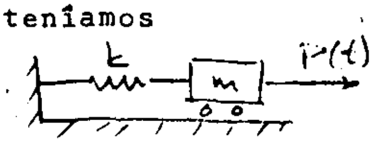
$$x_1(t) = r_{11}\, Y_1(t) + r_{12}\, Y_3(t) + r_{13}\, Y_3(t)$$

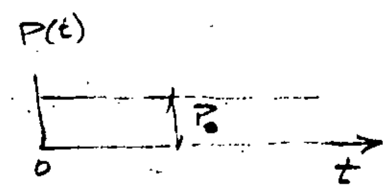
$$x_2(t) = r_{21}\, Y_1(t) + r_{22}\, Y_2(t) + r_{23}\, Y_3(t)$$

$$x_3(t) = r_{31}\, Y_1(t) + r_{32}\, Y_2(t) + r_{33}\, Y_3(t)$$

Otro ejemplo

Para 1GL teníamos





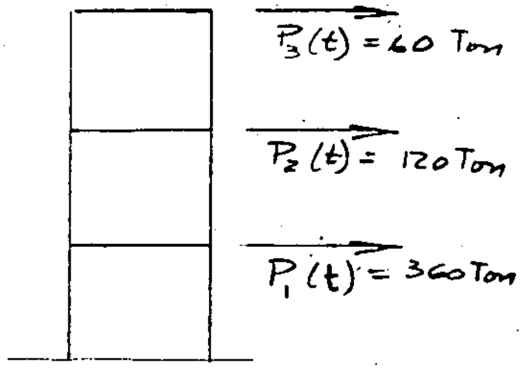
La ec:

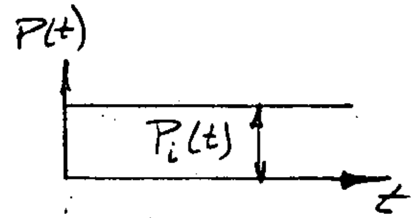
$$x + P^2 x = \frac{P(t)}{m} = \frac{P_o}{m}$$

y para CI = 0 la solución

$$x = \frac{P_o}{K}\left(1 - \cos pt\right)$$

Tenemos ahora el problema de encontrar la respuesta de





Para el modo j:

$$\ddot{Y}_j + p^2 Y_j = \frac{P_j^*(t)}{M_j^*} = \frac{P_{jo}^*}{M_j^*} \quad \text{cuya solución es:}$$

$$Y_j = \frac{P_{jo}^*}{K_j^*}\left(1 - \cos p_j t\right) = \frac{P_{jo}^*}{p_j^2\, M_j^*}\left(1 - \cos p_j t\right)$$

Cálculo de P_j^*

$$P_j^* = \{r\}_j' \; \{P(t)\} = \{r\}_j' \begin{Bmatrix} 360 \\ 120 \\ 60 \end{Bmatrix}$$

modo

$$\begin{array}{l|l} 1 & P_1^* = P_1 r_{11} + P_2 r_{21} + P_3\, r_{31} = 360+256.2+197.1 = 813.3 \\ 2 & P_2^* = P_1 r_{12} + P_2 r_{22} + P_3\, r_{32} = 360+107.88-88.4 = 379.48 \\ 3 & P_3^* = P_1 r_{13} + P_2\, r_{23} + P_3\, r_{33} = 360-125.28+24.66 = 259.98 \end{array}$$

Ahora bien,

$$Y_{j(st)} = \frac{P_j^*}{p_j^2\; M_j^*} = \frac{P_j^*}{K_j^*}$$

$$\therefore \quad Y_{1(st)} = \frac{813.30}{21 \times 19.629} = 1.973 \text{ cm}$$

$$Y_{2(st)} = \frac{379.48}{965 \times 5.386} = 0.730 \text{ cm}$$

$$Y_{3(st)} = \frac{259.38}{212.4 \times 3.804} = 0.321 \text{ cm}$$

de donde

$$Y_j = \frac{P_j^*}{p_j^2 M_j^*} \left(1 - \cos p_j t\right),$$ y tenemos:

$$Y_1(t) = Y_{1(st)} \left(1 - \cos p_1 t\right)$$

$$Y_2(t) = Y_{2(st)} \left(1 - \cos p_2 t\right)$$

$$Y_3(t) = Y_{3(st)} \left(1 - \cos p_3 t\right)$$

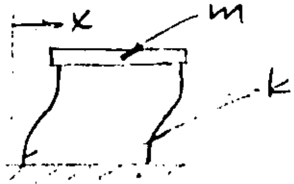
y, finalmente:

$$\{x(t)\} = \{r_1\} Y_1(t) + \{r_2\} Y_2(t) + \{r_3\} Y_3(t) = [R] \{Y\}$$

$$\begin{Bmatrix} X_1(t) \\ X_2(t) \\ X_3(t) \end{Bmatrix} = \begin{Bmatrix} 1.000 \\ 2.135 \\ 3.285 \end{Bmatrix} 1.973 \left(1-\cos p_1 t\right) + \ldots + \begin{Bmatrix} 1.000 \\ -1.044 \\ 0.411 \end{Bmatrix} 0.321 \left(1-\cos p_3 t\right)$$

EXCITACION SISMICA

A. Sistemas 1GL

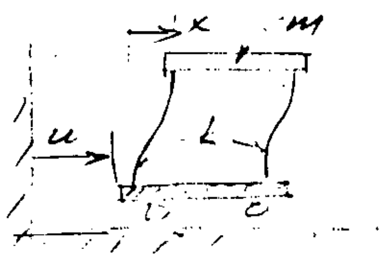


$$m\ddot{x} + kx + P(t) \qquad (a)$$

Para P(t) cualquiera y para CI ≠ 0 la solución de (a) es:

$$x(t) = x_o \cos pt + \frac{\dot{x}_o}{p} \operatorname{sen} pt + \frac{1}{mp}\int_0^t P(\tau) \operatorname{sen} p(t-\tau)\, d\tau$$

Para excitación sísmica:



$$m(\ddot{x} + \ddot{u}) + k\,x = 0$$

o sea,

$$m\ddot{x} + k\,x = -m\ddot{u} \qquad (b)$$

De la comparación de (a) y (b), la solución completa de ésta es:

$$x(t) = x_o \cos pt + \frac{\dot{x}_o}{p} \operatorname{sen} pt - \frac{1}{p}\int_0^t \ddot{u}(\tau) \operatorname{sen} p(t-\tau)\, d\tau$$

B. Sistemas de nGL:

$$[M]\{\ddot{x}\} + [k]\{x\} = \{P(t)\} = \begin{Bmatrix} P_1(t) \\ P_2(t) \\ \vdots \\ P_n(t) \end{Bmatrix} = \begin{Bmatrix} -m_1\ddot{u} \\ -m_2\ddot{u} \\ \vdots \\ -m_n\ddot{u} \end{Bmatrix}$$

$$= -\begin{Bmatrix} m_1 \\ m_2 \\ \vdots \\ m_n \end{Bmatrix}\ddot{u} = -\{m\}\ddot{u}$$

Es decir, tenemos:

$$[M]\{\ddot{x}\} + [K]\{x\} = \{P(t)\} = -\{m\}\ \ddot{u}$$

sust. $\{x\} = [R]\{y\}$

$$[M]\,[R]\,\{\ddot{y}\} + [K]\,[R]\,\{y\} = \{P(t)\} = -\{m\}\ \ddot{u}(t)$$

pre x $\{r\}'_j$

$$\{r\}'_j\,[M][R]\{\ddot{y}\} + \{r\}'_j\,[K][R]\{y\} = \underbrace{\{r\}'_j\,\{P\}}_{P_j^*} = \underbrace{-\underbrace{\{r\}'_j\,\{m\}}_{m_j^*}\ \ddot{u}}_{U_j^*}$$

por ortogonalidad:

$$\{r\}'_j\,[M]\{r\}\ \ddot{y}_j + \{r\}'_j\,[K]\,\{r\}_j\ y_j = P_j^* = U_j^*$$

y queda:

$$M_j^*\ \ddot{y}_j + K_j^*\ y_j = P_j^* = U_j^* = -\ m_j^*\ \ddot{u}$$

$\therefore$ la solución (CI = 0) de esta ecuación es:

Para P_j^* :

$$y_j(t) = \frac{1}{\phi_j M_j^*}\int_0^t P_j^*(\tau)\ \mathrm{sen}\ \phi_j(t-\tau)\,d\tau$$

Para U_J^*:

$$y_j(t) = \frac{1}{\phi_j M_j^*}\int_0^t U_j^*(\tau)\ \mathrm{sen}\ \phi_j(t-\tau)\,d\tau$$

que puede escribirse:

$$y_j(t) = -\frac{m_j^*}{p_j M_j^*}\int_0^t \ddot{u}(Z)\,\text{sen}\,p_j(t-Z)\,dZ$$

$$+\ y_{oj}\cos p_j t + \frac{\dot{y}_{oj}}{p_j}\ \text{sen}\,p_j t \qquad \begin{matrix}\text{términos a}\\ \text{para}\\ CI \neq 0\end{matrix}$$

Una vez obtenidos los elementos de $\{y\}$ solo falta premultiplicar por $[R]$ para obtener $\{x\}$:

$$\{x(t)\} = [R]\ \{y(t)\}$$

<u>GENERALIZACIÓN DE LAS CONDICIONES DE ORTOGONALIDAD</u>

Tenemos la ecuación:

$$\left[[K] - p^2\,[M]\right]\{x\} = \{0\}$$

que convenimos en escribir en la forma:

$$(K - p^2 M)\ x = 0$$

como los vectores modales la satisfacen:

$$K\,r_j = p_j^2\ M\,r_j \qquad \text{(a)}$$

y premultiplicando por: $r_i' \, MM^{-1}$ tenemos:

$$r_i'\ M\ M^{-1} \qquad K\,r_j = p_j^2\ M\ M^{-1} \qquad M\,r_j = p_j^2\ M\ M^{-1} K\,r_j = 0$$

que puede escribirse

$$r_i' \; M \; (M^{-1} \; K)^2 \; r_j = 0$$

y así podría seguirse para llegar a:

$$r_i' \; M \; (M^{-1}K)^{\ell} \; r_j = 0 \quad - \begin{cases} \ell \text{ entero} \\ -\infty < \ell < \infty \end{cases}$$

$$r_i' \; M \; (M^{-1} \; K)^{\ell} \; r_j = 0 \qquad \text{(b)}$$

en forma análoga podemos obtener

$$r_i' \; (MF)^{\ell} \; M \; r_j = 0 \qquad \text{(c)}$$

o

$$r_i' \; (K \; M^{-1})^{\ell} \; K \; r_j = 0$$

En (b):

$\ell = -2 \qquad M \; (M^{-1}K)^{-2} = M \; (M^{-1}K)^{-1} \; (M^{-1}K)^{-1}$

(en (c), con ℓ=2) $\qquad = M \; K^{-1} \; M \; K^{-1} \; M = \underline{M \; F \; M \; F \; M}$

$\ell = -1 \qquad M(M^{-1}K)^{-1} = M \; K^{-1} \; M = \underline{M \; F \; M}$

$\ell = 0 \qquad M \; (M^{-1}K)^0 = \underline{M}$

$\ell = 1 \qquad M \; (M^{-1}K)^1 = M \; M^{-1} \; K = \underline{K}$

$\ell = 2 \qquad M \; (M^{-1}K)^2 = M \; M^{-1} \; K \; M^{-1} \; K = \underline{K \; M^{-1} \; K}$

$\ell = 3 \qquad M \; (M^{-1}K)^3 = M \; M^{-1} \; K \; M^{-1} \; K \; M^{-1} K = \underline{K \; M^{-1} \; K \; M^{-1} \; K}$

VIBRACION LIBRE Y FORZADA DE SISTEMAS DE N GL CON AMORTIGUAMIENTO

Las ecuaciones de equilibrio dinámico son:

$$\{F_I\} + \{F_a\} + \{F_r\} = \{P(t)\}$$

Ya tenemos:

$$\{F_I\} = [M] \{\ddot{x}\}$$

$$\{F_r\} = [K] \{x\}$$

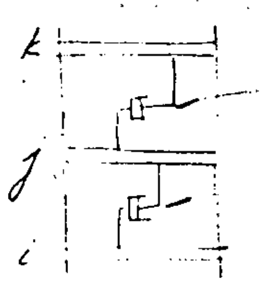
y ahora hacemos

$$\{F_a\} = [c] \{\dot{x}\}$$

donde

$$[c] = [c_{ij}]$$

y c_{ij} = fuerza de amortiguamiento en la coordenada i debido a una velocidad unitaria en la coordenada j.



$$\left.\begin{array}{l} c_{jk} = c_{kj} \\ \\ c_{ij} = c_{ji} \end{array}\right\} \text{indica acoplamiento}$$

La ecuación de movimiento es

$$[M] \{\ddot{x}\} + [c] \{\dot{x}\} + [K] \{x\} = \{P(t)\}$$

Hagamos: $\{X\} = [R]\{y\}$ premultiplicando por $\{r\}_j'$

$$\{r\}_j'[M][R]\{\ddot{y}\} + \{r\}_j'[C][R]\{\dot{y}\} + \{r\}_j'[K][R]\{y\} = \{r\}_j'\{P(t)\}$$

Para desacoplar estas ecuaciones debemos tener

$$\left.\begin{array}{ll} \{r\}_j'[M]\{r\}_i = 0 & i \neq j \\ \{r\}_j'[K]\{r\}_i = 0 & i \neq j \end{array}\right\} \text{cierto por ortogonalidad}$$

$$\{r\}_j'[C]\{r\}_i = 0 \quad i \neq j \qquad \text{¿pero ésta?} \quad (a)$$

1° <u>admitamos</u> que se cumple:

Ya definimos

$$\{r\}_j'[M]\{r\}_j = M_j^* \qquad \{r\}_j'\{P(t)\} = P_j^*$$

$$\{r\}_j'[K]\{r\}_j = K_j^*$$

y ahora

$$\{r\}_j'[C]\{r\}_j == C_j^* = 2\beta_j \not{p}_j M_j^*$$

y nuestra ecuación para el modo j queda:

$$M_j^* \ddot{y}_j + 2\beta_j \not{p}_j M_j^* \dot{y}_j + \not{p}_j^2 M_j^* y_j = P_j^*$$

o bien:

$$\ddot{y}_j + 2\beta_j \not{p}_j \dot{y}_j + \not{p}_j^2 y_j = \frac{P_j^*}{M_j^*}$$

Como las soluciones para un sistema de 1GL (cuya ec. es $\ddot{x}+2\beta p\dot{x}+p^2x = \frac{P(t)}{m}$) ya las conocemos, solo nos falta saber cómo debe ser $[C]$ para que se cumpla

$$\{r\}_i' [C] \{r\}_j = 0 \qquad i \neq j \tag{a}$$

además, claro, de

y $$\left.\begin{aligned} \{r\}_i' [M] \{r\}_j &= 0 \\ \{r\}_i' [K] \{r\}_j &= 0 \end{aligned}\right\} i \neq j$$

La ec. (a) se satisface si

i) $[C]$ es proporcionala $[M]$ o a $[K]$

ii) $[C]$ es una combinacion lineal de $[M]$ y $[K]$, o sea:

$$[C] = a_0 [M] + a_1 [K]$$

esto es muy restringido.

iii) En forma más general:

$$[C] = [M] \sum_{l} a_l [M^{-1}K]^l = \sum_{l} [C_l] \tag{38.1}$$

pues ya sabemos que todas las posibles formas $[M]\ [M^{-1}K]^l$ son satisfactorias y (38.1) es una C. L de matrices de este tipo.

La selección adecuada de a_l dará a $[C]$ las propiedades deseadas, o sea, podremos dar valores específicos a los elementos de $[C]$. ¿Cuáles le damos?

Asignamos un cierto valor de β a cada modo.

$$C_j^* = \{r\}_j' [C] \{r\}_j = 2\beta_j p_j M_j^* = \sum_l \{r\}_j' [C_l] \{r\}_j = \sum_l C_{jl}^* \quad (38.2)$$

De 38.1 y A

$$C_{jl}^* = \{r\}_j' [M] [M^{-1}K]^l \{r\}_j a_l \quad (38.3)$$

Por otra parte, para vibración libre:

$$(K - p_j^2 M) r_j = 0$$

$$K r_j = p_j^2 M r_j \leftrightarrow \frac{1}{p_j^2} r_j = F M r_j$$

premultiplicando por $r_j' M$:

$$\frac{1}{p_j^2} r_j' M r_j = r_j' M F M r_j$$

es decir

$$(p_j^2)^{-1} M_j^* = r_j' M (M^{-1}K)^{-1} r_j$$

y así podríamos llegar a que, para cualquier l:

$$(\phi_j^2)^l M_j^* = r_j' M(M^{-1}K)^l r_j = \underbrace{\frac{C_{jl}^*}{a_l}}_{\text{por } 38.3} \qquad 39.1$$

De 39.1:

$$C_{jl}^* = (\phi_j^2)^l M_j^* a_l$$

$$C_{jl}^* = (\phi_j^2)^l M_j^* a_l$$

y sumando sobre l:

$$\sum_l C_{jl}^* = \sum_l (\phi_j^2)^l M_j^* a_l$$

pero ya teníamos que

$$\sum_l C_{jl}^* = 2\beta_j \phi_j M_j^*$$

$$\therefore \qquad 2\beta_j \phi_j \cancel{M_j^*} = \sum_l (P_j^2)^l \cancel{M_j^*} a_l$$

de donde:

$$\boxed{\beta_j = \frac{1}{2\phi_j} \sum_l (\phi_j^2)^l a_l}$$

Con los n valores de β_j para los n modos podemos resolver para los n valores de a_l y formar nuestra $[C]$ con la ecuación

$$[C] = [M] \sum_{l} a_l \left[M^{-1}K\right]^{l}$$

Por ejemplo para nuestra estructura de 3GL asignemos:

$$\beta_1 = 0.10, \qquad \beta_2 = 0.05, \qquad \beta_3 = 0.02$$

$$\beta_1 = 0.10 = \frac{1}{2p_1}\left[a_1(p_1^2)^{-1}+a_0(p_1^2)^{0}+a_1(p_1^2)^{1}\right]$$

$$\beta_2 = 0.05 = \frac{1}{2p_2}\left[a_{-1}(p_2^2)^{-1}+a_0(p_2^2)^{0}+a_1(p_2^2)^{1}\right]$$

$$\beta_3 = 0.02 = \frac{1}{2p_3}\left[a_{-1}(p_3^2)^{-1}+a_0(p_3^2)^{0}+a_1(p_3^2)^{1}\right]$$

o, en forma matricial:

$$\begin{Bmatrix} 0.10 \\ 0.05 \\ 0.02 \end{Bmatrix} = \frac{1}{2}\begin{bmatrix} 1/p_1^3 & 1/p_1 & p_1 \\ 1/p_2^3 & 1/p_2 & p_2 \\ 1/p_3^3 & 1/p_3 & p_3 \end{bmatrix}\begin{Bmatrix} a_{-1} \\ a_0 \\ a_1 \end{Bmatrix}$$

al resolver para a_l resulta

$$\underline{[C] = a_{-1}\,[MFM] + a_0\,[M] + a_1\,[K]}$$

En p. tenemos que para CI = 0 y $\beta = 0$, para excitación sísmica

$$y_j(t) = -\frac{m_j^*}{P_j M_j^*}\int_0^t \ddot{u}_u(\tau)\,\text{sen}\,P_j(t-\tau)\,d\tau$$

coeficiente de participación = $\dfrac{m_j^*}{M_j^*}$

$$C_j = \frac{m_j^*}{M_j^*} = \frac{\{r\}_j^t \{m\}}{\{r\}_j^t [M] \{r\}_j} = \frac{\sum_{i=1}^{m} m_i r_{ij}}{\sum_{i=1}^{m} m_i r_{ij}^2}$$

y $\therefore$ podemos poner:

$$y_j(t) = C_j z_j(t)$$

en la que C_j está definida arriba y

$$z_j(t) = -\frac{1}{p_j}\int_0^t \ddot{u}(\tau)\,\mathrm{sen}\,p_j(t-\tau)\,d\tau$$

(y semejante si $\beta \neq 0$)

$$y_j(t) = C_j z_j(t)$$

Además, tenemos

$$\{X\} = [R]\{y\}$$

o sea

$$\begin{Bmatrix} X_1 \\ X_2 \\ \vdots \\ X_i \\ \vdots \\ X_n \end{Bmatrix} = \begin{bmatrix} r_{11} & r_{12} & \cdots & r_{1j} & \cdots & r_{1n} \\ r_{21} & r_{22} & \cdots & r_{2j} & \cdots & r_{2n} \\ \vdots & \vdots & & \vdots & & \vdots \\ r_{n1} & r_{n2} & \cdots & r_{nj} & \cdots & r_{nn} \end{bmatrix} \begin{Bmatrix} Y_1 \\ Y_2 \\ \vdots \\ Y_n \end{Bmatrix}$$

$$x_i = \sum_{j=1}^{n} r_{ij} y_j = \sum_{j=1}^{n} r_{ij} C_j z_j(t)$$

De aquí (sin sumar para todos los modos)

$$\left.\begin{aligned} |X_{ij}|_{max} &= r_{ij} C_j |z_j(t)|_{max} = r_{ij} C_j S_d \\ &= r_{ij} C_j \frac{S_a}{p_j^2} \end{aligned}\right\} S_a = pS_v = p^2 S_d$$

De esta ec. pasamos a:

$$|X_i|_{\substack{máx \\ ABS}} = \sum_{j=1}^{n} r_{ij} C_j S_d = \sum_{j=1}^{n} r_{ij} C_j \frac{S_a}{p_j^2}$$

$$|x_i|_{\substack{máx \\ PROB}} = \sqrt{\Sigma (|X_{ij}|_{máx})^2}$$

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

CURSOS ABIERTOS

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

MODULO II: ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMO

METODOS DE STODOLO-VIANELLO-NEWMARK Y DE HOLZER PARA EL CALCULO DE FRECUENCIAS Y CONFIGURACIONES MODALES

M. I. JOSE LUIS TRIGOS SUAREZ

METODOS DE STODOLA-VIANELLO-NEWMARK Y DE HOLZER PARA EL CALCULO DE FRECUENCIAS Y CONFIGURACIONES MODALES

ENRIQUE DEL VALLE C*

Para calcular las frecuencias y configuraciones modales de estructuras idealizadas como una serie de masas unidas por resortes, sin amortiguamiento, en vibración libre, se puede suponer que cada masa se mueve en movimiento armónico simple definido por $X=X_0 \cos wt$ o $X=X_0 \operatorname{sen} wt$ donde X_0 define la amplitud y w la frecuencia circular del movimiento.

La aceleración estará dada entonces por $\ddot{X}=-w^2X_0 \cos wt$ ó $\ddot{X}=-w^2X_0 \operatorname{sen} wt=-w^2X$ y las fuerzas de inercia a que estará sometida cada masa, de acuerdo con la segunda ley de Newton, serán $Fi = m\ddot{X} = -mw^2X$.

Por otro lado, la fuerza restitutiva que aparece en cada resorte estará dada por $Fe=R\Delta X$, donde R es la rigidez de entrepiso, que podemos definir como la fuerza cortante que es necesario aplicar para producir un desplazamiento unitario entre dos niveles consecutivos: $R = V/\Delta X$, para $\Delta X=1$.

Vemos entonces, que las fuerzas a que se verá sujeta cada masa dependerán de X y de w^2 únicamente.

Por otro lado, sabemos que para conocer un modo de vibrar necesitamos conocer tanto la frecuencia w (o periódo T) como la configuración modal relativa, y que si la estructura está vibrando en un modo dado, la frecuencia del movimiento de cada masa será la misma.

Tomando en cuenta lo anterior, se pueden emplear dos métodos numéricos para el cálculo de las frecuencias y configuraciones modales.

*Profesor Titular, División de Estudios de Posgrado, Fac. de Ingeniería UNAM.

El método propuesto por Stodola-Vianello-Newmark, consiste en:

1. Suponer una configuración deformada de la estructura:

   $X_{i\,supuesta}$

2. Valuar las fuerzas de inercia asociadas a esa configuración $F_i = -mw^2 X_i$, dejando w^2 como factor común cuyo valor no conocemos.

3. Valuar la fuerza cortante en la estructura, como la suma acumulativa de las fuerzas de inercia de arriba abajo del edificio: $V_i = \sum_{1=n}^{i} F_i$ (función de w^2)

4. Calcular los incrementos de deformación correspondientes a las fuerzas cortantes.

   $\Delta X_i = \dfrac{V_i}{R_i}$ (función de w^2).

5. Obtener la configuración calculada de la estructura como la suma acumulativa de los incrementos de deformación, de abajo hacia arriba.

   $$X_{i\,calc} = \sum_{i=1}^{n} \Delta X_i = \text{coef. } w^2$$

   Esto nos dará un coeficiente multiplicado por w^2 para cada masa.

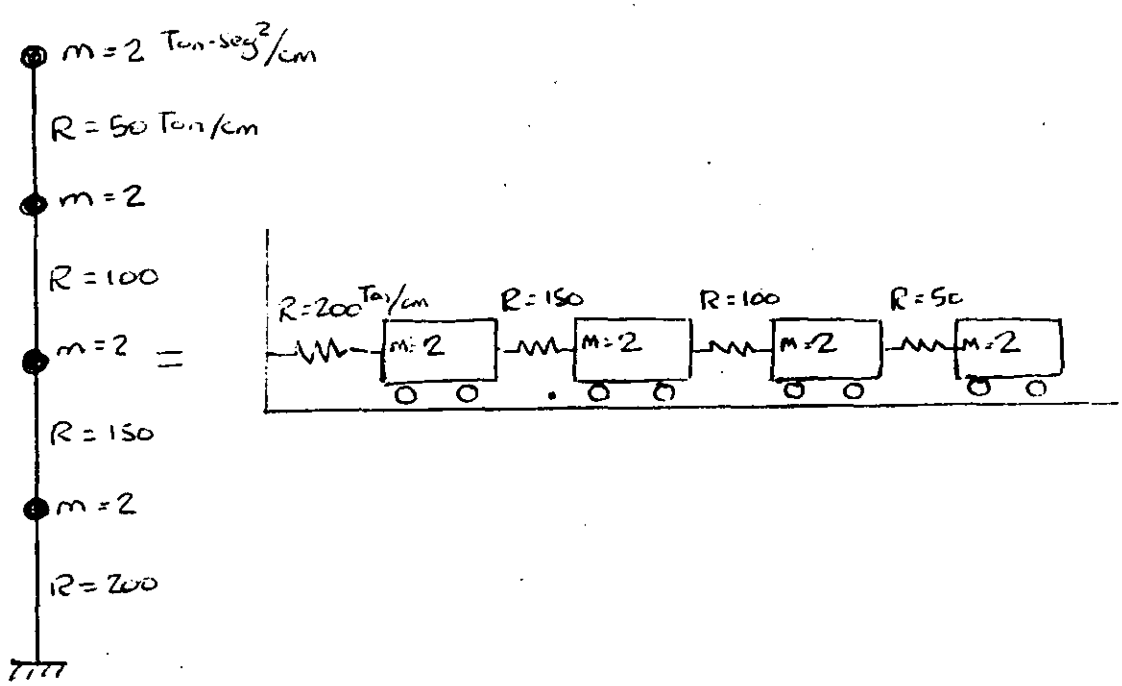
6. Si la estructura está vibrando en un modo la configuración calculada será proporcional a la supuesta, y el factor de proporcionalidad será w^2. Esto es, para cada masa podremos calcular.

   $$w^2 = \frac{X_{supuesta}}{\text{Coef. de } X_{calc.}}$$

   En general, los valores de w^2 calculados para cada masa, no serán iguales en el primer ciclo, pero el método es de rápida convergencia si se usa como nueva configuración supuesta la obtenida al final de cada ciclo, de preferencia normalizándola, esto es, haciendo que la deformación de una de las masas, por ejemplo la primera, tenga siempre el mismo valor, con objeto de observar como se modifica la configuración relativa después de cada ciclo. Los valores de w^2 obtenidos en cada ciclo nos dan también un intervalo de valores que se va cerrando hasta que se obtiene finalmente los mismos valores para todas las masas.

El método descrito anteriormente converge siempre hacia el modo más bajo que esté presente en la configuración supuesta, y dado que al suponer una configuración ésta estará formada por una combinación lineal de todos los modos posibles, el modo más bajo será el primero o fundamental. Más adelante se indica como hacer para calcular modos superiores.

Ejemplo. Calcular la frecuencia y configuración modal del primer modo de vibrar de la estructura representada por el modelo matemático siguiente.



Para realizar los pasos antes indicados conviene usar una tabulación como la siguiente:

1er. Ciclo.

Nivel	$\frac{\text{ton seg}^2}{\text{cm}}$ m	$\frac{\text{ton}}{\text{cm}}$ R	cm* X_{sup}	$F_i = m w^2 X$	V	$\Delta X = \frac{V}{R}$	X_{calc}	** w^2	*** X_{sup}
4	2		4	$8w^2$			$0.52w^2$	$7.692 = \frac{4}{0.52}$	5.2
		50			$8w^2$	$0.16w^2$			
3	2		3	$6w^2$			$0.36w^2$	$8.333 = \frac{3}{0.36}$	3.6
		100			$14w^2$	$0.14w^2$			
2	2		2	$4w^2$			$0.22w^2$	$9.091 = \frac{2}{0.22}$	2.2
		150			$18w^2$	$0.12w^2$			
1	2		1	$2w^2$			$0.1w^2$	$10.0 = \frac{1}{0.1}$	1
		200			$20w^2$	$0.1w^2$			
0							0		

Nótese que los valores R, V y ΔX están defasados, pues corresponden al entrepiso.

* Para iniciar el cálculo puede usarse cualquier valor de X. En general, el método convergirá más rápido entre más acertada sea la configuración supuesta, pero si se supone por ejemplo una configuración que se parezca a un segundo, tercero o cuarto modo, de cualquier manera, al término de algunos ciclos más, llegaremos al primer modo.

** Nótese que en este caso, el valor de w^2 estará comprendido entre

$$7.692 \frac{1}{\text{seg}^2} \quad \text{y} \quad 10 \frac{1}{\text{seg}^2}$$

*** En un segundo ciclo, usaremos como nueva configuración supuesta la obtenida al final del primer ciclo normalizada de tal modo que la deformación del primer nivel, sea unitaria, esto es, dividiendo la configuración calculada entre $0.1w^2$ en cada nivel.

2o. Ciclo

Nivel	m	R	x_{sup}	Fi	V	ΔX	X	w^2	x_{sup}
4	2		5.2	$10.4w^2$			$0.651w^2$	7.988	5.425
		50			$10.4w^2$	$0.208w^2$			
3	2		3.6	$7.2w^2$			$0.443w^2$	8.126	3.692
		100			$17.6w^2$	$0.176w^2$			
2	2		2.2	$4.4w^2$			$0.267w^2$	8.240	2.225
		150			$22w^2$	$0.147w^2$			
1	2		1.0	$2.\ w^2$			$0.120w^2$	8.333	1.0
		200			$24w^2$	$0.120w^2$			
0							0		

Obsérvese que el intervalo de variación de w^2 se redujo a 7.988 y 8.333 y que las variaciones en la configuración modal fueron mucho menores que las que tuvo el primer ciclo.

Tomando como base de partida nuevamente la configuración calculada, en un tercer ciclo se tiene:

Nivel	m	R	x_{sup}	F	V	ΔX	X	w^2	X_i
4	2		5.425	$10.85w^2$			$0.6739w^2$	8.050	5.461
		50			$10.85w^2$	$0.2170w^2$			
3	2		3.692	$7.384w^2$			$0.4569w^2$	8.081	3.703
		100			$18.234w^2$	$0.1823w^2$			
2	2		2.225	$4.45w^2$			$0.2746w^2$	8.103	2.225
		150			$22.684w^2$	$0.1512w^2$			
1	2		1.0	$2.0\ w^2$			$0.1234w^2$	8.104	1.00
		200			$24.684w^2$	$0.1234w^2$			
0							0		

y finalmente, en un cuarto ciclo, la aproximación se considera suficiente:

Nivel	m	R	X_{sup}	F	V	ΔX	X_{calc}	w^2	Xi
4	2	50	5.461	$10.922w^2$	$10.922w^2$	$0.2184w^2$	$0.6775w^2$	8.061	5.468
3	2	100	3.703	$7.406w^2$	$18.328w^2$	$0.1833w^2$	$0.4591w^2$	8.066	3.705
2	2	150	2.225	$4.45w^2$	$22.778w^2$	$0.1519w^2$	$0.2758w^2$	8.067	2.226
1	2	200	1.00	$2.00w^2$	$24.778w^2$	$0.1239w^2$	$0.1239w^2$	8.071	1.00
0		Σ	12.389			$\Sigma =$	$1.5363w^2$	8.064*	

*El valor final de w^2 lo obtenemos con más precisión dividiendo la suma de X_{sup} entre la suma de coeficientes de X_{calc} Esto es más preciso que promediar los valores de w^2 de cada nivel.

$$w=\sqrt{8.064} = 2.8397; \quad T= \frac{2\pi}{w} = \frac{6.2832}{2.8397} = 2.213 \text{ seg.}$$

Cálculo de modos superiores empleando este método

Como se indicó antes, el método converge al modo más bajo presente en la configuración supuesta, y al suponer una combinación cualquiera ésta, estará constituida por una combinación lineal de los distintos modos de vibrar:

$X_{sup} = C_1X_{i1}+C_2Xi_2+C_3X_{i3}+C_4X_{i4}$, donde X_{i1} a X_{i4} son las configuraciones modales y C_i son coeficientes de participación.

Si queremos calcular el segundo modo de vibrar empleando este método, tendremos que quitar a la configuración supuesta la participación del primer modo: C_1X_{i1}, para lo cual necesitamos conocer X_{i1} y C_1. X_{i1} la calculamos como se indicó antes y C_1 lo podemos calcular recurriendo a la propiedad de ortogonalidad de los modos de vibración que indica que $\Sigma m_i X_{in} X_{im}=0$ si $n\neq m$, donde X_{in} y X_{im} son configuraciones modales.

Si multiplicamos la expresión anterior de X_{sup} por $m_i X_{i1}$ y sumamos para todas las masas, considerando que los coeficientes de participación son constantes y pueden salir de la sumatoria, tendremos:

$$\Sigma m_i X_{i1} X_{sup} = C_1 \Sigma m_i X_{i1}^2 + C_2 \Sigma m_i X_{i1} X_{i2} + C_3 \Sigma m_i X_{i1} X_{i3} + \ldots$$

donde los términos que multiplican a C_2, C_3, etc. son nulos por la propiedad de ortogonalidad de los modos, quedando entonces

$$C_1 = \frac{\Sigma m_i X_{i1} X_{sup}}{m_i X^2{}_{i1}}$$

Esta expresión es válida para cualquier modo n.

Por tanto, si queremos calcular el segundo modo de vibrar, supondremos una configuración que se parezca a este modo, es decir, que tenga un punto de deflexión nula, calcularemos el valor de C_1 con la expresión anterior y restaremos a la configuración supuesta para el segundo modo la participación del primer modo $C_1\ X_{i1}$, lo que da por resultado una nueva configuración supuesta para el segundo modo en la que el modo más bajo presente es el segundo y por lo tanto, al aplicar el método habrá convergencia hacia este modo. A la operación antes descrita se le llama "limpia" de modos.

Si quisiéramos calcular el tercer modo de vibrar, tendríamos que conocer de antemano las configuraciones correctas de primero y segundo modo, y suponer una configuración que se parezca al tercer modo, (que tenga dos puntos de deflexión nula); calcularíamos dos coeficientes de participación C_1 y C_2, correspondientes a los modos primero y segundo, en la configuración supuesta y la limpiaríamos para que el modo más bajo presente en ella sea el tercero y el método converja a este modo.

Esto es:

$$X_{i3sup} = C_1 X_{i1} + C_2 X_{i2} + C_3 X_{i3} + C_4 X_{i4} + \ldots$$

$$C_1 = \frac{\Sigma m X_{i1} X_{i3sup}}{\Sigma m X_{i1}^2} \quad ; \quad C_2 = \frac{\Sigma m X_{i2} X_{i3sup}}{\Sigma m X_{i2}^2}$$

$$\overline{X}_{i3sup} = X_{i3sup} - C_1 X_{i1} - C_2 X_{i2} = C_3 X_{i3} + C_4 X_{i4} + \ldots$$

De manera semejante se procede para calcular otros modos superiores.

En la práctica, y debido a errores numéricos o de aproximación que van acarréandose no basta con una sola limpia. Para lograr convergencia adecuada da buen resultado limpiar la configuración calculada al cabo de cada ciclo, antes de calcular los valores de w^2. Esa misma configuración limpiada, normalizada, nos sirve como nueva configuración para un nuevo ciclo. Es conveniente llevar cuando menos tres cifras significativas en los cálculos.

Para fijar ideas, calcularemos tres ciclos del segundo modo de vibrar de la estructura para la cual calculamos anteriormente el primer modo.

Nivel	m	R	X_{i1}	mX^2_{i1}	mX^2_{i1}	X_{i2sup}*	$mX_{i1}X_{i2sup}$	$-C_1 X_{i1}$	$\overline{X}_{i2}$	$F_{i2} = mX_{i2}w^2$	V	ΔX	X Calc.
4	2		5.468	10.936	59.798	-1.0	-10.936	-0.054	-1.054	$-2.108w^2$			$-0.0334w^2$
		50									$-2.108w^2$	$-0.0422w^2$	
3	2		3.705	7.41	27.454	0	0	-0.036	-0.036	$-0.072w^2$			$0.0088w^2$
		100									$-2.180w^2$	$-0.0218w^2$	
2	2		2.226	4.452	9.910	2.0	9.910	-0.022	1.978	$3.956w^2$			$0.0306w^2$
		150									$1.776w^2$	$0.0118w^2$	
1	2		1.00	2.0	2.0	1.0	2.0	-0.010	0.990	$1.980w^2$			$0.0188w^2$
		200									$3.756w^2$	$0.0188w^2$	
0				Σ =	99.162		Σ = 0.974						

DATOS

$C_1 = \frac{0.974}{99.162}$ 0.00982

*La configuración supuesta puede ser cualquiera, pero desde luego en conveniente que se parezca a un segundo modo, esto es, que tenga un cambio de signo en la configuración modal.

[illegible]	$mx_{i1}X_{calc}$	$-C_1 Xi_1$	$\bar{X}_{2calc}$	w^2	X_{i2sup} **	$m\bar{X}_{i2sup}w^2$	V	ΔX	X_{calc}
4	$-0.3653w^2$	$+0.00696w^2$	$-0.02644w^2$	39.86	-1.3042	$-2.6084w^2$	$-2.6084w^2$	$-0.05217w^2$	$-0.0314w^2$
3	$0.0652w^2$	$0.00472w^2$	$0.01352w^2$	-2.66	0.6669	$1.3338w^2$	$-1.2746w^2$	$-0.01275w^2$	$0.02077w^2$
2	$0.1362w^2$	$0.00284w^2$	$0.03344w^2$	59.15	1.6495	$3.2990w^2$	$2.0244w^2$	$0.01350w^2$	$0.0335w^2$
1	$.0376w^2$	$0.00127w^2$	$0.02007w^2$	49.33	0.990	$1.9800w^2$	$4.0044w^2$	$0.02002w^2$	$0.02002w^2$
Σ	$-0.1263w^2$								

$$C_1 = \frac{-0.1263w^2}{99.162} = -0.0012736w^2$$

** Normalizando con respecto a 0.99 en el primer nivel, para comparar la evolución de la configuración.

Nivel	$mX_{i1}X_{cal}$	$-C_1 X_{i1}$	$\bar{X}_{2\ cal}$	w^2***	$\bar{X}_{i2sup}$	$m\bar{X}_{i2sup}w^2$	V	ΔX
4	$-0.34339w^2$	$+0.000012w^2$	$-0.031388w^2$	41.55	-1.5520	$-3.104w^2$	$-3.104w^2$	$-0.06208w^2$
3	$0.15391w^2$	$+0.000008w^2$	$0.020778w^2$	32.10	1.0274	$2.0548w^2$	$-1.0492w^2$	$-0.01049w^2$
2	$0.14923w^2$	$+0.000005w^2$	$0.033525w^2$	49.20	1.6577	$3.3154w^2$	$2.2662w^2$	$0.01511w^2$
1	$0.04004w^2$	$+0.000002w^2$	$0.020022w^2$	49.45	0.99	$1.98w^2$	$4.2462w^2$	$0.02123w^2$

$\Sigma = -0.00021w^2$

$\Sigma = 2.1231$

$\Sigma\ |\,|\ 5.2271$

$$C_1 = \frac{-0.00021w^2}{99.162} = -0.0000021177w^2$$

*** Nótese que el intervalo de w^2 queda comprendido entre 32.1 y 49.49 y que el ajuste en la curva ocurre casi entre las dos últimas masas. Obsérvese que la corrección al limpiar es muy pequeña.

Nivel	X_{calc}	$mX_{i1}X_{calc}$	$-C_1X_{i1}$	$\bar{X}_{calc}$	w^2 ****	$\bar{X}_{isup}$
4	$-0.03623w^2$	$-0.39621w^2$	+0.000023	$-0.036207w^2$	42.86	-1.705
3	$0.02585w^2$	$0.19155w^2$	+0.000015	$0.025865w^2$	39.72	1.206
2	$0.03634w^2$	$0.16179w^2$	+0.000009	$0.036349w^2$	45.61	1.695
1	$0.02123w^2$	$0.04246w^2$	+0.000004	$0.021234w^2$	46.62	0.99
0	0	$\Sigma -0.00041w^2$		$\Sigma = 0.047241w^2$	prom. 43.70 44.94	
				$\Sigma = 0.119655w^2$ (vals. abs)	43.68	

$$C_i = \frac{-0.00041w^2}{99.162} = -0.0000041w^2$$

****El intervalo de variación de w^2 se ha reducido a 39.72 - 46.62 (dif. = 6.9) y los ajustes en la curva son menores. En uno o dos ciclos más se llegaría al valor correcto de w^2 y X_i. Nótese que para estimar un valor de w^2 procediendo como se indicó anteriormente podemos hacer las sumas de $\bar{X}_{sup}$ y de los coeficientes de $\bar{X}_{calc}$ tomando valores absolutos o tomando en cuenta el signo correspondiente. La variación que se obtiene en este caso es de 3% aprox. Si sacamos el promedio de w^2 se obtiene un valor casi igual al obtenido con las sumas de valores absolutos, que es más correcto.

Si no hubiéramos hecho la limpia en ninguno de los ciclos, al cabo de 8 habríamos llegado a la configuración del primer modo (en vez de 4 ciclos que se necesitaron cuando la configuración supuesta se parecía a la del primer modo).

Aplicación del Método de Stodola-Vianello-Newmark para Estructuras de Flexión

Como se verá más adelante, cuando las trabes de los marcos son muy flexibles en comparación con las columnas, o cuando las fuerzas laterales son resistidas por muros que trabajan esencialmente a flexión, la rigidez de entrepiso no es independiente de la distribución de fuerzas a que esté sometida la estructura y por tanto no puede suponerse constante para el cálculo de los distintos modos de vibrar. En general, la pseudorigidez equivalente que se obtendría para un segundo modo será mayor que la correspondiente al primer modo, pues los efectos de flexión de conjunto se reducen considerablemente al no tener todas las fuerzas actuando en el mismo sentido. Lo mismo podría decirse para modos superiores (ref. 1).

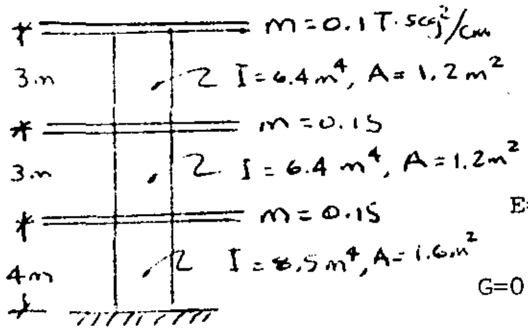
En esos casos, las propiedades elástico geométricas de la estructura no quedarán definidas por rigideces de entrepiso sino por la variación de los productos EI y GA con los cuales se podrán calcular las deformaciones debidas a flexión y a fuerza cortante respectivamente.

Para calcular las deformaciones por flexión es conveniente el empleo de los teoremas de la viga conjugada, que es, para el caso de un voladizo, otro voladizo empotrado en el extremo opuesto cargado con el diagrama de momentos entre EI, y en el cual los momentos flexionantes corresponden a las deformaciones de la viga real.

Las deformaciones por cortante, que en el caso de estructuras a base de muros pueden ser importantes en comparación con las de flexión, sobre todo en los niveles inferiores, se calculan mediante la expresión $\Delta X_{v_i} = \frac{V_i h_i}{A_i G}$, donde ΔX_{v_i} es el incremento de deformación por cortante entre dos niveles consecutivos, V_i, h_i y A_i son, respectivamente la fuerza cortante, la altura y el área efectiva de cortante entre esos mismos niveles y G es el módulo de elasticidad al cortante del material de la estructura.

Para calcular los modos de vibración, se supone una configuración modal, se calculan las fuerzas de inercia $F_i = m_i w^2 X_i$ asociadas a la configuración y las fuerzas cortantes correspondientes y a partir de ellas se valúan los incrementos de momento de cada entrepiso y los momentos de volteo acumulados de arriba hacia abajo, los cuales se dividen entre EI (habrá dos valores de M/EI en un mismo nivel en los casos en que haya cambio de sección de los muros). La integración numérica del diagrama de M/EI nos permitirá transformar ese diagrama en una serie de cargas concentradas equivalentes a él aplicadas en los distintos niveles con los cuales es muy fácil calcular los cortantes equivalentes correspondientes a cada entrepiso y los incrementos de momento flexionante en la viga conjugada que serán iguales a los incrementos de deformación por flexión entre dos niveles consecutivos (es el equivalente de $\Delta X = V/R$ del caso visto anteriormente). A estos incrementos de deformación por flexión se sumarán los correspondientes a la deformación por cortante y con esa suma se podrá calcular la nueva configuración, que será como antes función de w^2 y de donde podremos despejar este valor y en caso de que no sea igual para todas las masas volver a hacer otro ciclo tomando como configuración de partida la encontrada anteriormente normalizándola con respecto a una de las masas para poder comparar la evolución de las configuraciones de cada ciclo.

Para fijar ideas, a continuación se presenta un ejemplo de análisis de una estructura en que las fuerzas laterales son resistidas por muros, cuyos valores de I y A son los indicados en la figura siguiente:



E=200 000 kg/cm^2=2 000 000 Ton/m^2=2x10^6 Ton/m^2

G=0.4 E=0.8x10^6 Ton/m^2

Ni-vel	$\frac{\text{T-seg}^2}{\text{cm}}$ m	m^4 I	Ton-m^2 EI	m^2 A	Ton GA	m h	X_{sup}^{cm}	$mX_{sup}\,w^2$	Ton V	ΔM=Vh	Ton-m M	1/m $\frac{M}{EI}$
3	0.10	6.4	12.8x10^6	1.2	0.96x10^6	3	5.0	0.50w^2	0.5w^2	1.5w^2	0	0
2	0.15	6.4	12.8x10^6	1.2	0.96x10^6	3	2.5	0.38w^2	0.88w^2	2.64w^2	1.5w^2	0.1172x10^{-6}w^2
1	0.15	8.5	17.0x10^6	1.6	1.20x10^6	4	1	0.15w^2	1.03w^2	4.12w^2	4.14w^2	0.3234x10^{-6}w^2 0.2435x10^{-6}w^2
											8.26w^2	0.4859x10^{-6}w^2

Ejemplo de cálculo de las concentraciones equivalentes al diagrama de M/EI

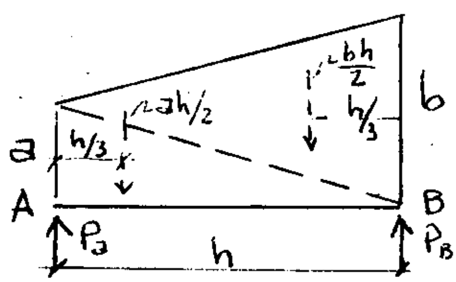
Para el nivel 3

$$P_{eq} = \frac{3}{6}\ (2x0 + 0.1172 \times 10^{-6}\ w^2) \ = 0.0586x10^{-6}w^2$$

(Ver aclaración al pie de la tabla de la página siguiente)

Nivel	Peq*	Veq**	$\Delta M = Veq\ h = \Delta X_f^m$	ΔX_v^{m***}	ΔX_{tot}^m	X_{cal}^m
3	$0.0586\times10^{-6}w^2$					$23.0052\times10^{-6}w^2$
		$2.2369\times10^{-6}w^2$	$6.7107\times10^{-6}w^2$	$1.5625\times10^{-6}w^2$	$8.2732\times10^{-6}w^2$	
	$0.1172\times10^{-6}w^2$					
2						$14.732\times10^{-6}w^2$
	$0.2789\times10^{-6}w^2$					
		$1.8408\times10^{-6}w^2$	$5.5224\times10^{-6}w^2$	$2.75\times10^{-6}w^2$	$8.2724\times10^{-6}w^2$	
	$0.3820\times10^{-6}w^2$					
1						$6.4596\times10^{-6}w^2$
	$0.6486\times10^{-6}w^2$					
		$0.8102\times10^{-6}w^2$	$3.2408\times10^{-6}w^2$	$3.2188\times10^{-6}w^2$	$6.4596\times10^{-6}w^2$	
0	$0.8102\times10^{-6}w^2$					0

$1/seg^2$ w^2 ****	x_{sup}
2173.42	3.56
1696.99	2.28
1548.08	1.0



* Para obtener cargas concentradas equivalentes al diagrama de M/EI se puede usar la fórmula siguiente:

$$P_a = \frac{h}{6}(2a+b); \quad P_b = \frac{h}{6}(2b+a)$$

donde h es la distancia entre dos puntos A y B con ordenadas de M/EI iguales a a y b respectivamente. La variación de M/EI entre A y B es lineal, por lo que esta expresión se obtiene considerando dos triángulos con alturas a y b respectivamente y base h. Pa y Pb son las concentraciones correspondientes en los puntos A y B. (Ref. 2).

** Recuérdese que el empotramiento de la viga conjugada es el extremo superior, por lo que empieza de abajo hacia arriba el cálculo.

***Obsérvese que en el primer entrepiso la deformación por cortante es prácticamente igual a la de flexión por lo que despreciarla conduciría a errores muy grandes. Al ir aumentando la altura de la estructura la deformación por cortante va reduciendo su importancia en comparación con la de flexión y puede llegar a ser despreciable. En este caso la deformación por cortante en el tercer entrepiso es 23% de la debida a flexión.

**** Debe tenerse cuidado con las unidades al valuar w^2 pues es fácil equivocarse, obsérvese que X_{sup} está en cm y X calc resulta en metros.

Método de Holzer

Como se indicó anteriormente, para conocer completamente un modo de vibrar necesitamos conocer tanto la configuración modal como la frecuencia del modo. Hemos visto que en el método Stodola-Vianello-Newmark se supone una configuración relativa y a partir de ella se calcula el valor de w^2. Holzer procede exactamente alrevés, esto es, supone la frecuencia y a partir de ella se calcula la configuración relativa de abajo hacia arriba de la estructura. Dado que la configuración es relativa se puede suponer también la deformación de la primera masa (por consiguiente el incremento de deformación entre la base y la primera masa). El método tiene las siguientes etapas:

Los datos son las masas y las rigideces de entrepiso, igual que antes.

1. Suponer un valor de w^2.
2. Obtener los valores de mw^2_{sup} para cada masa.
3. Suponer la deformación del primer nivel: X_1; conviene suponer un valor unitario. Esto equivale también, como ya se dijo a suponer ΔX_i.

4. Calcular la fuerza cortante en la base de la estructura, (Primer entrepiso) que será por definición de rigidez de entrepiso:

   $V_1 = R_1 \Delta X_1 \quad \text{si} \quad \Delta X_1 = 1, \; V_1 = R_1$

5. Calcular la fuerza de inercia asociada a la masa del primer nivel:

   $F_1 = m_1 w^2_{sup} X_1$

6. Por definición de fuerza cortante, como la suma acumulativa de las fuerzas arriba de un cierto nivel, podremos calcular la cortante del segundo entrepiso restando a la cortante en la base la fuerza de inercia del primer nivel, esto es:

   $V_2 = V_1 - F_1$

7. Conocida la fuerza cortante en el entrepiso 2 podemos calcular el incremento de deformación en ese entrepiso dividiendo la cortante entre la rigidez de entrepiso $\Delta X_2 = \frac{V_2}{R_2}$

8. Sumando X_2 a la deformación del primer nivel obtendremos la deformación del segundo nivel $X_2 = X_1 + \Delta X_2$ y podemos repetir los pasos 5 a 8 para todas las masas hasta llegar al extremo superior de la estructura.

Si la frecuencia supuesta corresponde a un modo de vibrar, obtendremos que la fuerza de inercia del último nivel es igual a la fuerza cortante del entrepiso correspondiente (por equilibrio dinámico). Si la frecuencia supuesta no es la correspondiente a un modo de vibrar, se obtendrá una diferencia entre el valor de la fuerza de inercia y el de la fuerza cortante en el extremo de la estructura. En este caso el método no es convergente, pero si hacemos otro ciclo con otro valor de w^2 relativamente cercano al anterior, encontraremos otra diferencia y podremos trazar una gráfica que nos relacione las frecuencias supuestas (abscisas) con las diferencias entre fuerza de inercia y fuerza cortante en el extremo superior de la estructura (ordenadas). Una vez que tenemos dos puntos de esa gráfica podremos buscar un valor de w^2 supuesto en la intersección con el eje de las abscisas de la línea que une los puntos antes obtenidos, o su prolongación si ambas diferencias tienen el mismo signo. Con este tercer valor supuesto para w^2 seguramente obtendremos otra diferencia, menor que las anteriores, que nos definirá un tercer punto en la gráfica. Podremos entonces trazar una curva entre los tres puntos y definir así un nuevo valor de w^2 que seguramente estará muy próximo a la frecuencia correcta de uno de los modos de vibrar de la estructura.

Cuando ya se está cerca del valor correcto, se puede mejorar el valor supuesto de w^2 empleando el cociente de Crandall siguiente:

$$\bar{w}^2 = w^2 \frac{\Sigma V \ \Delta X}{\Sigma F X}$$

donde $\bar{w}^2$es el valor que debemos suponer en el ciclo siguiente.

El método presentado sirve para calcular cualquier modo natural de vibración teniendo como datos las masas y las rigideces de entrepiso de la estructura. El modo de que se trate se obtendrá de la inspección de la configuración modal, tomando en cuenta que en el primero todas las deformaciones tienen el mismo signo, en el segundo hay un cambio de signo, en el tercero dos cambios de signo y así sucesivamente.

Si se conoce la frecuencia del primer modo de vibrar (por haberlo calculado empleando el método Stodola-Vianello-Newmark, por ejemplo), se puede estimar gruesamente el valor de las frecuencias de los modos superiores empleando la relación $w_2^2 \doteq 9w_1^2$; $w_3^2 \doteq 25w_1^2$, etc.

(Esta aproximación puede ser demasiado burda dependiendo de los valores relativos de las masas y rigideces en cada caso particular, pero sirve como orientación).

Ejemplo:

Calculemos el segundo modo de vibrar de la estructura que se usó en el método de Stodola-Vianello-Newmark, suponiendo

$$w_2^2 \doteq 9w_1^2 = 9 \times 8 = 72 \ \frac{1}{\text{seg}}{}^2$$

Usaremos la tabulación siguiente:

Ni-vel	m	R	mw^2_{sup}	ΔX	X*	F	V	
4	2		144		-2.751	-396.1		Dif = 260.7
		50		-2.707			-135.4	
3	2		144		-0.044	- 6.3		
		100		-1.417			-141.7	
2	2		144		1.373	-197.7		
		150		0.373			56	
1	2		144		$\boxed{1.0}$	144		
		200		$\boxed{1.0}$			200	

$$\boxed{w^2_{sup} = 72}$$

*Obsérvese que aunque la diferencia encontrada es fuerte, la configuración se parece a un segundo modo, pues tiene un cambio de signo.

Usando un nuevo valor de w^2_{sup} de $50 \times 1/seg^2$, tendremos

Ni-vel	m	R	mw^2_{sup}	ΔX	X	F	V	
4	2		100		-2.334	-233.4	Dif. 66.7	
		50		-3.334			-166.7	
3	2		100		1.00	100		
		100		-0.667			-66.7	
2	2		100		1.667	166.7		
		150		0.667			100	
1	2		100	1.00	1.0	100		
		200					200	

Trazando la gráfica w^2_{sup} -diferencias encontramos:

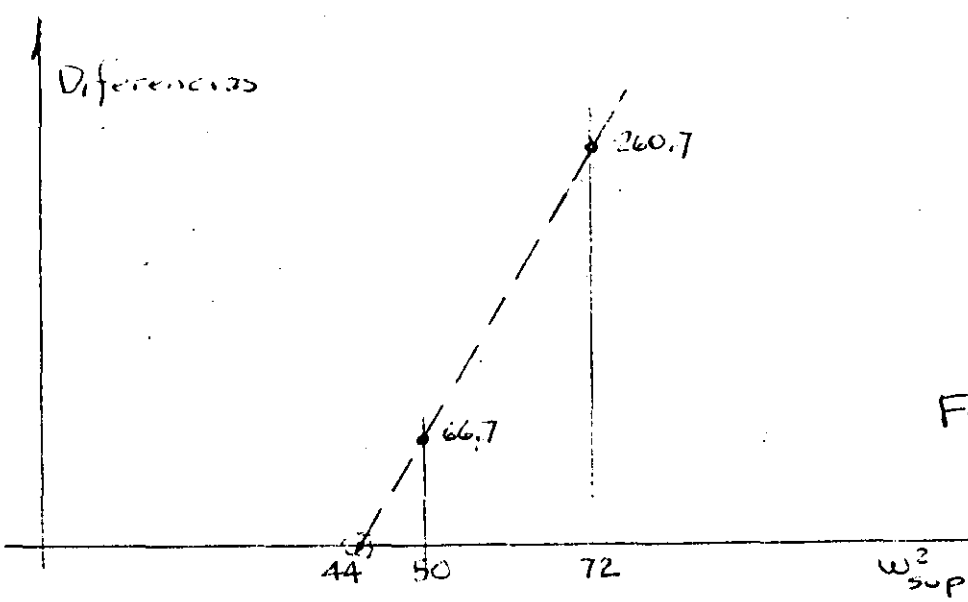


Fig 1.

que el valor de w^2 que hace cero las diferencias es aproximadamente 44 (podría obtenerse por triángulos semejantes, pero sabemos que aún cuando se hiciera así el valor no nos llevará exactamente a cero diferencia pues la variación no es lineal como estamos suponiendo, excepto en intervalos muy cerrados).

Suponiendo entonces $w^2 = 44$

Ni-vel	m	R	mw^2	ΔX	X	F	V	FX	$V\Delta X$
							Dif.=3.57		
4	2		88		-1.844	-162.27		299.23	
		50		-3.174			-158.7		503.71
3	2		88		1.33	117		155.61	
		100		-0.417			- 41.7		17.39
2	2		88		1.747	153.7		268.51	
		150		0.747			112		83.66
1	2		88		1.0	88		88	
		200		1.0			200		200
0								Σ811.35	804.76

$$\bar{w}^2 = 44\ \frac{804.76}{811.35} = 43.64\ 1/seg^2$$

Usando $w^2_{sup} = 43.64\ \ 1/seg^2$

Ni-vel	m	R	mw^2	ΔX	X	F	V
4	2		87.28		-1.809	-157.89	Dif. = 0.05
		50		-3.159			-157.94
3	2		87.28		1.350	117.83	
		100		-0.401			- 40.11
2	2		87.28		1.751	152.83	
		150		0.751			112.72
1	2		87.28		1.0	87.28	
		200		1.0			200
0							

Como puede verse, la diferencia al final de este último ciclo es despreciable, por lo que:

$$w_2^2 = 43.64\ 1/seg^2,\ w_2 = 6.606\ 1/seg.\quad T_2 = 0.951\ seg$$

y la configuración modal es la indicada.

Suponiendo otro valor mayor que w^2 podría calcularse el tercero y cuarto modos. Puede también verificarse que la frecuencia del primer modo obtenido con el método Stodola-Vianello-Newmark es correcta.

Comentarios adicionales

En los métodos presentados para las estructuras a base de marcos rígidos se tiene como datos las masas y las rigideces de entrepiso. Las masas son relativamente fáciles de calcular y dependen exclusivamente del peso de los materiales con que esté hecha la estructura y de la carga viva que se considere para fines de análisis sísmico. Las rigideces serán función de las propiedades elástico-geométricas de los materiales empleados, que no es sencillo definir y de la estructuración, sobre todo de la relación que guarden las rigideces relativas de las barras que forman la estructura, trabes y columnas.

Dado el modelo matemático de un edificio como una serie de masas unidas por resortes, definimos como sistema estrechamente acoplado a aquel en que la rigidez de entrepiso es independiente de la distribución de cargas laterales a que se vea sometido el modelo, esto es, la rididez de entrepiso es invariable independientemente de la elástica que adquiera la estructura al ser sometida a cargas laterales. Aquí se entiende por rigidez de entrepiso, como se indicó anteriormente, la fuerza necesaria para producir el desplazamiento unitario de un nivel con respecto al otro, esto es

$$R = \frac{V}{\Delta X}\quad ;\quad \text{para}\quad \Delta X=1,\quad R=V$$

En la figura 2 se muestra el modelo matemático de un edificio de 4 niveles sometido a distintos sistemas de fuerzas. De acuerdo con lo antes dicho, la rigidez debe ser independiente de las fuerzas aplicadas (este tipo de estructuras se conoce también como estructura "de cortante").

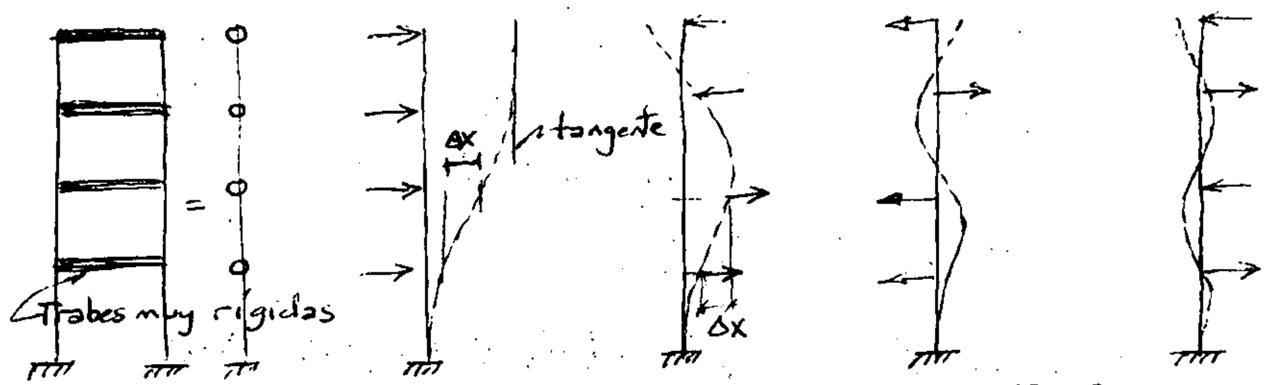
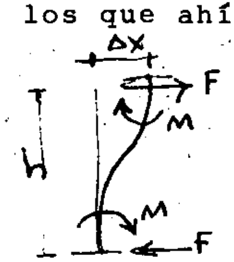


Fig 2

Para que esto se cumpla, la rigidez de entrepiso debe ser función única y exclusivamente de las columnas de cada entrepiso, para lo cual, los giros de los nudos deber ser nulos, lo que se logra si las trabes son infinitamente rígidas en comparación con las columnas, en cuyo caso la elástica de cada una de las columnas es la mostrada en la figura 3, y los elementos mecánicos que aparecen son los que ahí se muestran, para barras de sección constante.



$$F = \frac{12EI\Delta X}{h^3}$$

$$M = \frac{6EI\Delta X}{h^2}$$

Fig. 3

En la práctica, es difícil que la rigidez relativa de las trabes ($K=I/\ell$) sea muy grande en comparación con la de las columnas, lo que hará que los giros de los nudos no sean cero, relajándose el sistema y reduciéndose la rigidez del marco para un mismo tamaño de columnas. Debido a esto, el caso de trabes infinitamente rígidas en comparación con las columnas recibe a veces el nombre de cota superior de rigidez.

Al ser significativos los giros de los nudos, la rigidez de entrepiso ya no será independiente del sistema de fuerzas horizon tales aplicadas. En el límite inferior, llegaremos al caso del voladizo mostrado en la figura 4, para el cual no tiene sentido hablar de rigidez de entrepiso, pues será diferente para cada una de las posibles configuraciones de fuerzas aplicadas. A este caso lo definiremos como sistema remotamente acoplado.

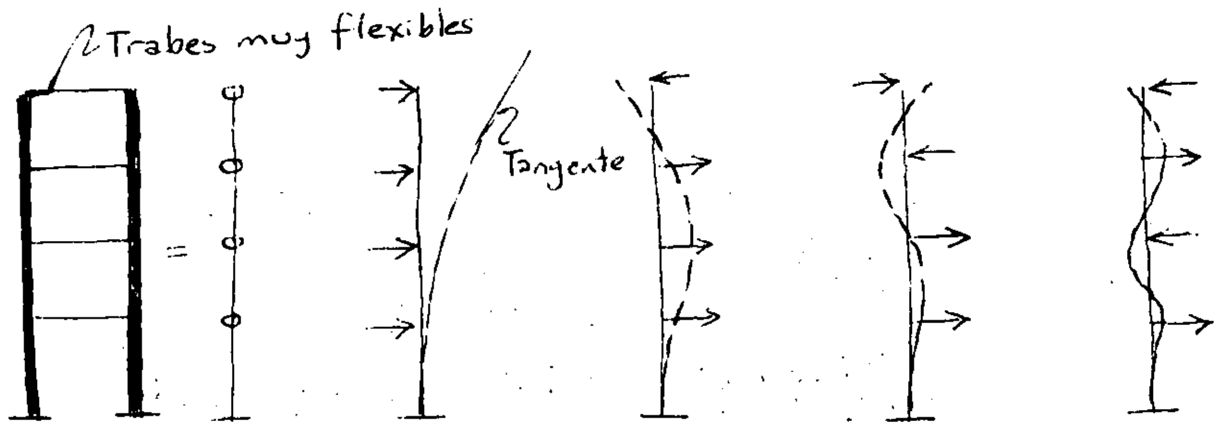
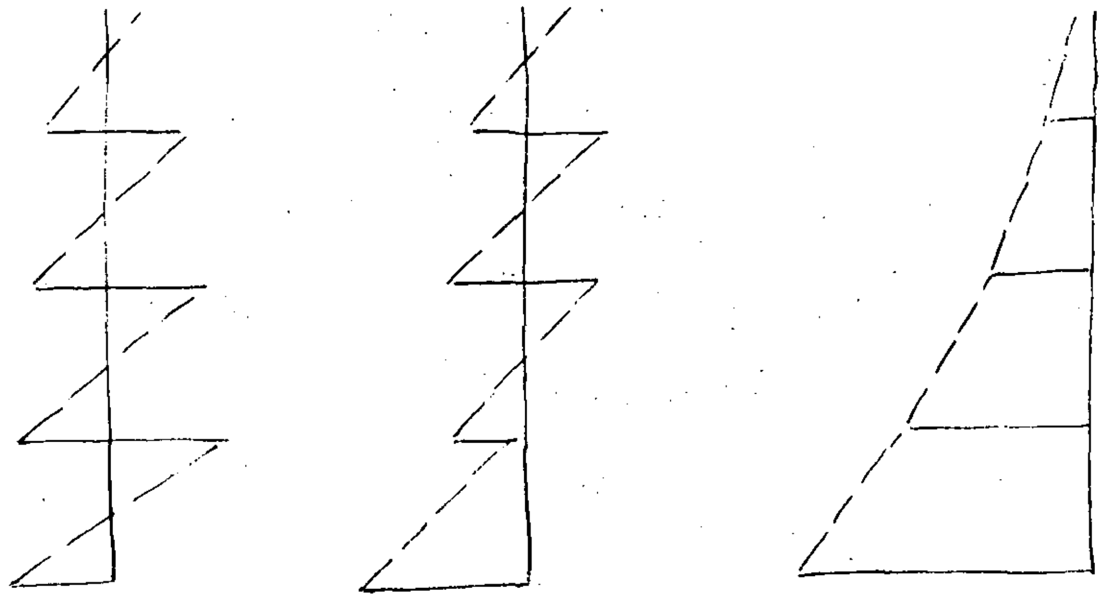


Fig 4

Nótese que en ambos casos se trata de estructuras aparentemente iguales, constituídas por marcos rígidos formados por trabes y columnas unidos en los nudos, sin embargo, como puede apreciarse en las figuras 1 y 3, las deformaciones de la estructura cuando todas las fuerzas se aplican en el mismo sentido son muy diferentes en uno y otro caso. En la figura 2, la tangente en el extremo superior es vertical, mientras que en la figura 4, la tangente en el extremo superior tiene la inclinación máxima.

La figura 5 ilustra la forma en que variarían los momentos flexionantes en las columnas del marco en los casos extremos y en uno intermedio. Nótese que la aplicación de métodos aproximados para la obtención de momentos en trabes y columnas sin verificar cual es la situación del marco, puede conducir a errores muy importantes de subestimación de momentos en las columnas y de desplazamientos horizontales de la estructura.



marco con trabes rígidas en comparación con las columnas.

marco en situación intermedia.

voladizo (trabes muy flexibles comparadas con las columnas).

Momentos flexionantes en columnas. Fig. 5

ya que los métodos aproximados en general suponen la formación de articulaciones (puntos de momento nulo) en cada entrepiso, y la situación puede ser tal que los puntos de inflexión del diagrama de momentos desaparezcan en uno, varios o todos los niveles.

Cualquier edificio de la práctica estará en una posición intermedia con respecto a los casos descritos.

Para conocer cual es la situación en cada caso particular, John A. Blume (referencia 1) sugiere el empleo de un indice de rotación nodal, que define como:

$$\rho = \frac{\Sigma(I/\ell)_{\text{trabes}}}{\Sigma(I/\ell)_{\text{cols}}}$$

y se puede valuar en cualquier entrepiso. (Blume lo hace para el entrepiso medio). Aquí $\Sigma(I/\ell)_{\text{trabes}}$ es la suma de rigideces relativas de las trabes de un cierto nivel y $\Sigma(I/\ell)_{\text{cols}}$ es la suma de rigideces relativas de las columnas en que se apoyan las trabes antes mencionadas.

Blume encontró que si $\rho>0.10$ hay puntos de momento nulo en las columnas de todos los entrepisos mientras que, para valores de ρ menores de 0.01 la estructura se asemeja más a un voladizo. Para valores de ρ entre 0.01 y 0.10 la situación es intermedia y habrá entrepisos en que no haya puntos de momento nulo, por lo que los métodos aproximados de análisis pueden conducir a fuertes errores del lado de la inseguridad por lo que respectá a los valores de los momentos flexionantes para los que debe diseñarse así como respecto a los desplazamientos laterales de la estructura; la rigidez de entrepiso pierde significado y conviene emplear métodos matriciales para analizarla.

Si la estructura tiene variaciones importantes con la altura, convendrá valuar ρ en distintos niveles.

Efectos de deformación axial de las columnas

Hasta aquí se ha considerado que las deformaciones axiales de las columnas, en el caso de marcos rígidos, son despreciables y no contribuyen a la deformación horizontal. Esto es válido sólo si la relación entre altura y ancho de la estructura es pequeña, tal vez menor que 3. Al aumentar el valor de esa relación, el efecto de momento de volteo en el edificio adquiere mayor importancia y se pueden cometer errores importantes al despreciar los acortamientos y alargamientos de las columnas debido a fuerza axial.

Cuando las trabes se vuelven muy flexibles en comparación con las columnas, cada una de las columnas trabajará como voladizo y la fuerza axial en ellas será pequeña.

En el caso de marcos contraventeados, la crujía o crujías contraventeadas tendrán comportamiento similar al de un muro y deberán por tanto considerarse como estructuras de flexión, calculando sus periodos como se indicó en el método Stodola-Vianello-Newmark.

Cuando se tienen marcos y muros trabajando simultáneamente la situación se complica pues la interacción entre ambos sistemas estructurales hace que varíe la fuerza que toman uno y otro en cada entrepiso; los muros suelen tomar la mayor proporción de la cortante total en los entrepisos inferiores mientras que la situación se invierte en los niveles superiores. Ver referencia 1. Esto hace difícil la aplicación de métodos numéricos para calcular los modos de vibración de este tipo de estructuras, siendo más conveniente el empleo de métodos matriciales para este fin.

REFERENCIA 1

Blume, John., "Dynamic Characteristics of Multistory Buildings", Procedings ASCE, Structural Division, February 1968.

REFERENCIA 2

Godden, William G., "Numerical Analysis of Beam and Column Structures", Prentice Hall.

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

C U R S O S   A B I E R T O S

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

MOD. II ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMO
Del 5 al 10 de junio de 1995.

ANALISIS SISMICO DE SISTEMAS DE UN GRADO DE LIBERTAD .

DR. OCTAVIO A. RASCON CHAVEZ.

Palacio de Minería Calle de Tacuba 5 Primer piso Deleg. Cuauhtémoc 06000 México, D.F. APDO. Postal M-2285
Teléfonos: 512-8955 512-5121 521-7335 521-1987 Fax 510-0573 521-4020 AL 26

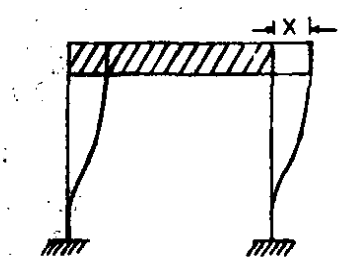
DINAMICA ESTRUCTURAL

DR. OCTAVIO A. RASCON CH.

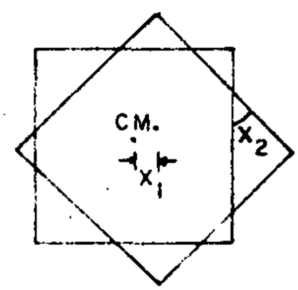
DEFINICION.

GRADOS DE LIBERTAD = NUMERO DE COORDENADAS GENERALIZADAS (DESPLAZAMIENTOS O GIROS) QUE SE REQUIEREN PARA DEFINIR LA POSICION DEL SISTEMA EN CUALQUIER INSTANTE.

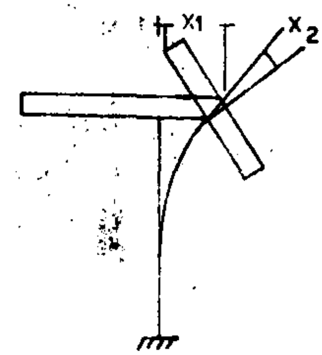
EJEMPLOS



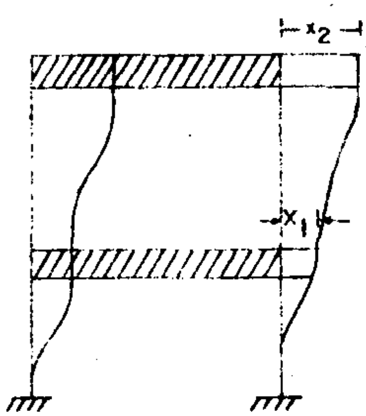
UN GRADO DE LIBERTAD



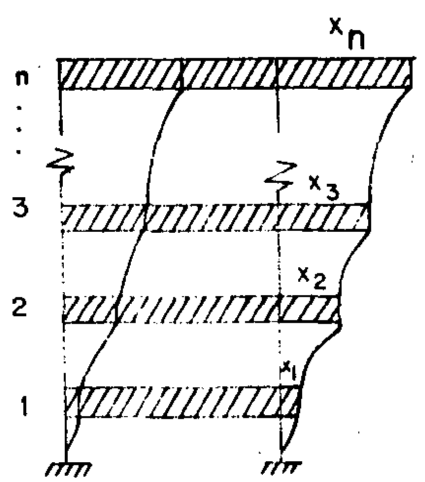
DOS GRADOS DE LIBERTAD



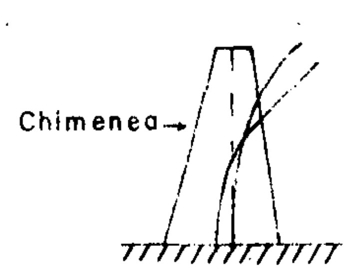
DOS GRADOS DE LIBERTAD



DOS GRADOS DE LIBERTAD



n GRADOS DE LIBERTAD



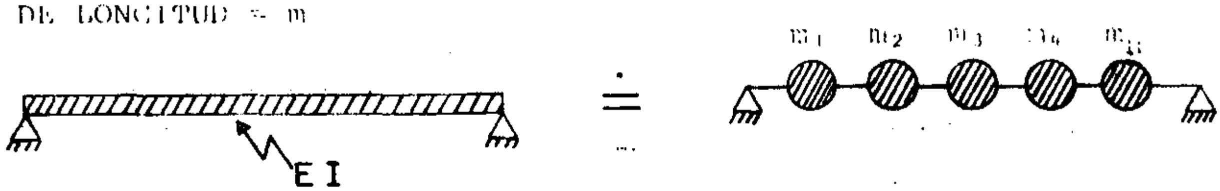
INFINITO NUMERO DE GRADOS DE LIBERTAD

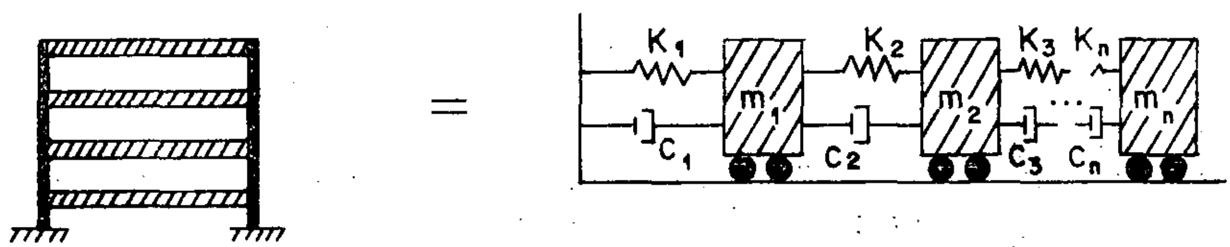
METODOS DE DISCRETIZACION DE SISTEMAS CONTINUOS

1. POR CONCENTRACION DE MASAS

MASA POR UNIDAD

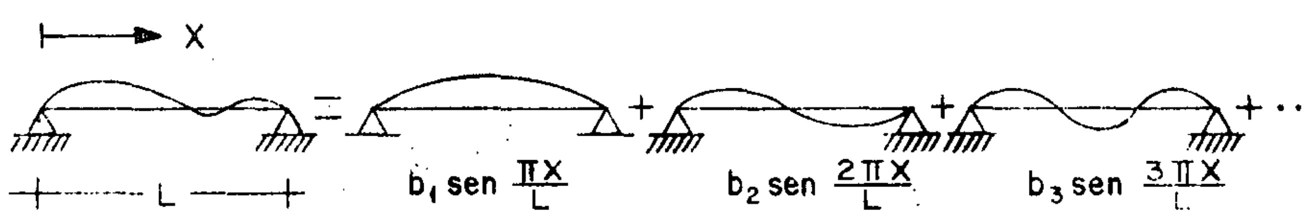
DE LONGITUD = m





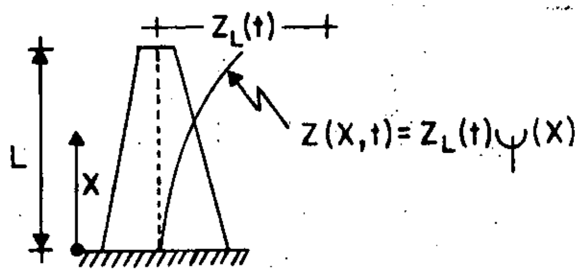
2. EXPRESANDO LA CONFIGURACION DE VIBRACION DE LA ESTRUCTURA COMO UNA SERIE DE FUNCIONES ESPECIFICADAS. POR EJEMPLO, SI ESTAS FUNCIONES SON ARMONICAS:

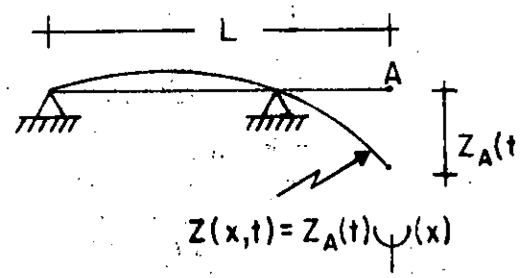
$$Z(x,t) = \sum_{i=1}^{N} b_i \operatorname{sen} \frac{i\pi x}{L}$$



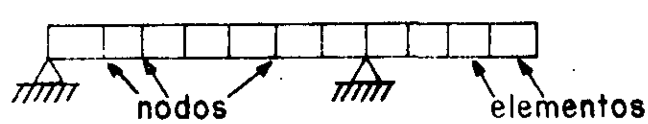
EN GENERAL, PARA CUALQUIER TIPO DE FUNCION $\psi(x)$:

$$z(x,t) = \sum_{i=1}^{N} z_i(t)\,\psi_i(x)$$

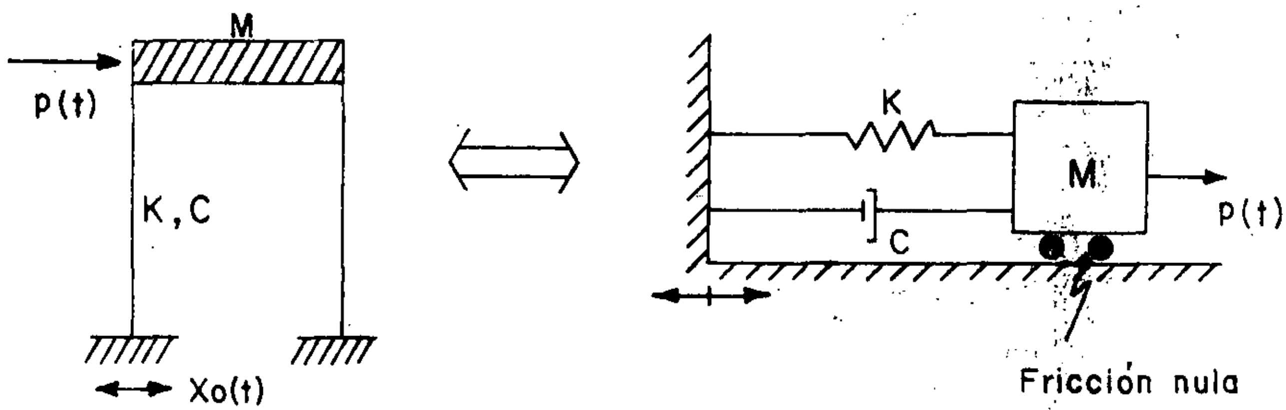




MEDIANTE ELEMENTOS FINITOS



RESPUESTA DINAMICA DE SISTEMAS ELASTICOS LINEALES DE UN GRADO DE LIBERTAD CON AMORTIGUAMIENTO VISCOSO



t = TIEMPO

M = MASA

K = RIGIDEZ

C = AMORTIGUAMIENTO

$f(t)$ = FUERZA EXTERNA

$X_o(t)$ = DESPLAZAMIENTO DEL SUELO

EL AMORTIGUAMIENTO VISCOSO ES TAL QUE PRODUCE UNA FUERZA DE RESTAURACION PROPORCIONAL A LA VELOCIDAD RELATIVA DE LA MASA RESPECTO AL SUELO.

EL AMORTIGUAMIENTO SE DEBE PRINCIPALMENTE A LA FRICCION INTERNA ENTRE LOS GRANOS O PARTICULAS DEL MATERIAL DE LA ESTRUCTURA, Y A FRICCION EN LAS JUNTAS Y CONEXIONES DE LA MISMA. ES EL ELEMENTO DEL SISTEMA QUE DISCIPA ENERGIA.

2a. LEY DE NEWTON:

"LA RAPIDEZ DE CAMBIO DEL MOMENTUM DE CUALQUIER MASA, m, ES IGUAL A LA FUERZA QUE ACTUA SOBRE ELLA"

$$p(t) = \frac{d}{dt}\left(m\frac{dx}{dt}\right) = \frac{d}{dt}(m\dot{x})$$

$p(t)$ = FUERZA ACTUANTE

x = DESPLAZAMIENTO

t = TIEMPO

SI m ES CONSTANTE: $p(t) = m\ddot{x}$

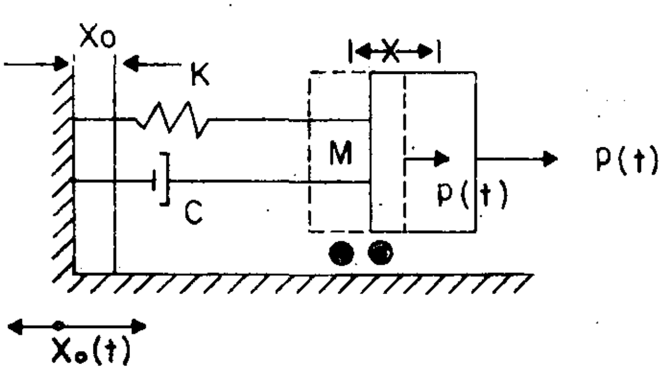
PRINCIPIO DE D'ALAMBERT

SI LA 2a. LEY DE NEWTON LA ESCRIBIMOS COMO

$$p(t) - m\ddot{x} = 0$$

AL SEGUNDO TERMINO DE LA ECUACION SE LE CONOCE COMO *FUERZA DE INERCIA*; EL CONCEPTO DE QUE UNA MASA DESARROLLA UNA FUERZA DE INERCIA PROPORCIONAL A SU ACELERACION Y QUE SE OPONE A ELLA SE CONOCE COMO PRINCIPIO DE D'ALAMBERT, Y PERMITE QUE LAS ECUACIONES DE MOVIMIENTO SE EXPRESEN COMO ECUACIONES DE *EQUILIBRIO DINAMICO.*

ECUACION DE EQUILIBRIO



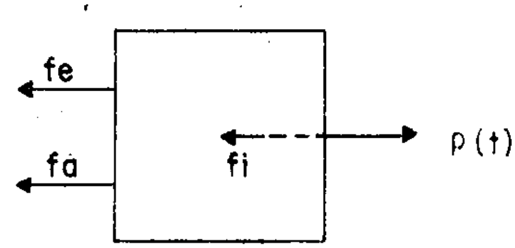


DIAGRAMA DE CUERPO LIBRE

EQUILIBRIO: $f_e + f_a + f_i = p(t)$ (1)

PARA UN SISTEMA ELASTICO: $f_e = K(x - x_o) = ky$

PARA AMORTIGUAMIENTO VISCOSO: $f_a = c(\dot{x} - \dot{x}_o) = C\dot{y}$ (2)

POR EL PRINCIPIO DE D'ALAMBERT

SUSTITUYENDO LAS ECS. 2 EN LA EC. 1 SE OBTIENE:

$$m(\ddot{y} + \ddot{x}_o) + c\dot{y} + k\dot{y} = p(t)$$

DE DONDE

$$\boxed{M\ddot{y} + c\dot{y} + Ky = p(t) - M\ddot{x}_o} \qquad (3)$$

DIVIDIENDO ENTRE M AMBOS MIEMBROS DE LA EC. 3:

$$\ddot{y} + \frac{C}{M}\dot{y} + \frac{K}{M}y = \frac{p(t)}{M} - \ddot{x}_o$$

SI $\frac{C}{M} = 2h$, y $\frac{K}{M} = \omega^2$, DONDE ω = FRECUENCIA CIRCULAR NATURAL, EN RAD/SEG:

$$\boxed{\ddot{y} + 2h\dot{y} + \omega^2 y = \frac{p(t)}{M} - \ddot{x}_o} \qquad (4)$$

CUANDO SE TIENEN EXCITACIONES EN EL SISTEMA SE TRATA DE UN PROBLEMA DE VIBRACIONES FORZADAS; EN CASO CONTRARIO EL PROBLEMA ES DE VIBRACIONES LIBRES.

VIBRACIONES LIBRES

EN ESTE CASO LA ECUACION DIFERENCIAL DE EQUILIBRIO RESULTA SER

$$\ddot{y} + 2h\dot{y} + \omega^2 y = 0$$

CUYA SOLUCION ES

$$y(t) = e^{-ht}(C_1 \text{ sen } \omega' t + C_2 \cos \omega' t) \qquad (5)$$

DONDE $\omega' = \sqrt{\omega^2 - h^2}$ = FRECUENCIA CIRCULAR NATURAL AMORTIGUADA

Y C_1 Y C_2 SON CONSTANTES QUE DEPENDEN DE LAS CONDICIONES INICIALES

(EN $t=0$) DE DESPLAZAMIENTO Y VELOCIDAD QUE TENGA LA MASA DEL SISTEMA.

ESTAS RESULTAN SER

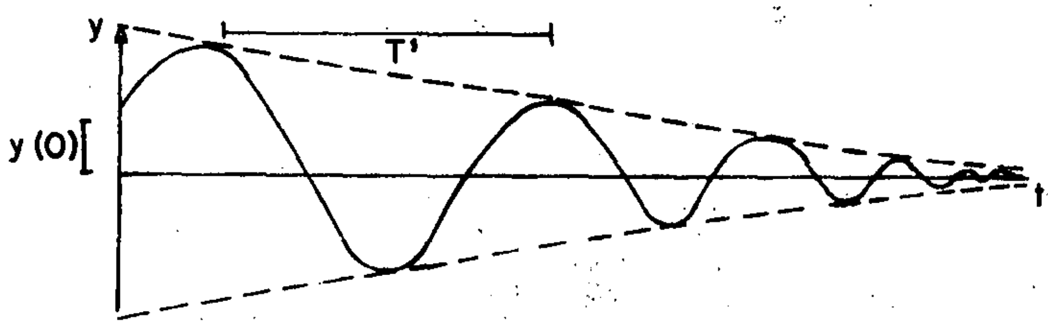
$$\boxed{C_1 = \frac{\dot{y}(0) + hy(0)}{\omega'}} \quad Y \quad \boxed{C_2 = y(0)} \tag{6}$$

LA EC (5) SE PUEDE ESCRIBIR TAMBIEN COMO:

$$y(t) = Ae^{-ht} \cos(\omega' t - \theta) \tag{7}$$

DONDE $A = \sqrt{C_1^2 + C_2^2}$ Y $\theta = \tan^{-1} \frac{C_1}{C_2}$ = ANGULO DE FASE

LA GRAFICA DE LA EC (7) ES



$T' = \frac{2\pi}{\omega'}$ = PERIODO NATURAL AMORTIGUADO, SEG

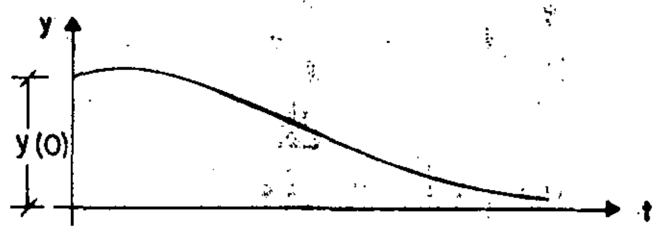
$f' = \frac{1}{T'}$ = FRECUENCIA NATURAL AMORTIGUADA, cps

VEAMOS EL CASO ESPECIAL DE LA EC. (5) EN QUE $h \to \omega$. EN TAL CASO, $\omega' = \sqrt{\omega^2 - h^2} \to 0$, $\cos \omega' t \to 1$ Y $\text{sen}\ \omega' t \to \omega' t$, CON LO CUAL LA EC. (5) SE REDUCE A

$$y(t) = e^{-\omega t} \{ [(\dot{y}(0) + hy(0))/\omega'](\omega' t) + y(0) \}$$

$$= e^{-\omega t} [\dot{y}(0) t + (1 + \omega t) y(0)]$$

LA GRAFICA DE ESTA ECUACION ES



Y OBVIAMENTE NO REPRESENTA UN MOVIMIENTO OSCILATORIO, POR LO CUAL SI $h = \omega$ SE DICE QUE SE TIENE AMORTIGUAMIENTO CRITICO. EN TAL CASO:

$$h_{cr} = \omega = \frac{C_{cr}}{2M} = \sqrt{\frac{K}{M}}$$

DE DONDE $\quad C_{cr} = 2\sqrt{KM}$ (8)

A LA RELACION $\zeta = C/C_{cr}$ SE LE LLAMA FRACCION DEL AMORTIGUAMIENTO CRITICO.

DESPEJANDO A M DE LA EC. (8) Y SUSTITUYENDOLA EN LA EC. $h = C/(2M)$ SE OBTIENE:

$$h = \frac{C}{2\,\frac{C_{cr}^2}{4K}} = \frac{C}{C_{cr}}\;\frac{2K}{2\sqrt{KM}} = \zeta\sqrt{\frac{K}{M}} = \zeta\omega$$

ADEMAS:

$$\omega' = \sqrt{\omega^2 - h^2} = \sqrt{\omega^2 - \omega^2\zeta^2} = \omega\sqrt{1-\zeta^2}$$

$$\omega' = \omega\sqrt{1 - \zeta^2} \qquad (9)$$

LOS VALORES USUALES EN ESTRUCTURAS QUE ASUME ζ VARIAN ENTRE 2 Y 5%. EN ESTE INTERVALO ω' Y ω SON CASI IGUALES; VEAMOS, POR EJEMPLO, EL CASO EN QUE $\zeta = 0.1$

$$\omega' = \omega\sqrt{1 - 0.01} = 0.995\omega$$

OTRA FORMA DE MEDIR EL GRADO DE AMORTIGUAMIENTO QUE TIENE UNA ESTRUCTURA ES MEDIANTE EL <u>DECREMENTO LOGARITMICO</u>, EL CUAL SE DEFINE COMO EL LOGARITMO DEL COCIENTE DE DOS AMPLITUDES CONSECUTIVAS

$$L = \ln \frac{y(t)}{y(t + T')} = \ln \frac{Ae^{-ht}\cos(\omega' t-\theta)}{Ae^{-h(t+T')}\cos[\omega'(t+T')-\theta]}$$

$$= \ln\left\{\frac{e^{-ht}}{e^{-h(t+T')}} \quad \frac{\cos(\omega' t - \theta)}{\cos(\omega' t + \omega' T' - \theta)}\right\}$$

$$= \ln\left\{\frac{e^{-ht}}{e^{-ht}\, e^{-hT'}} \quad \frac{\cos(\omega' t - \theta)}{\cos(\omega' t - \theta + 2\pi)}\right\}$$

$$= \ln e^{+hT'} = hT' = \zeta\omega T' = \zeta\omega \frac{2\pi}{\omega\sqrt{1-\zeta^2}}$$

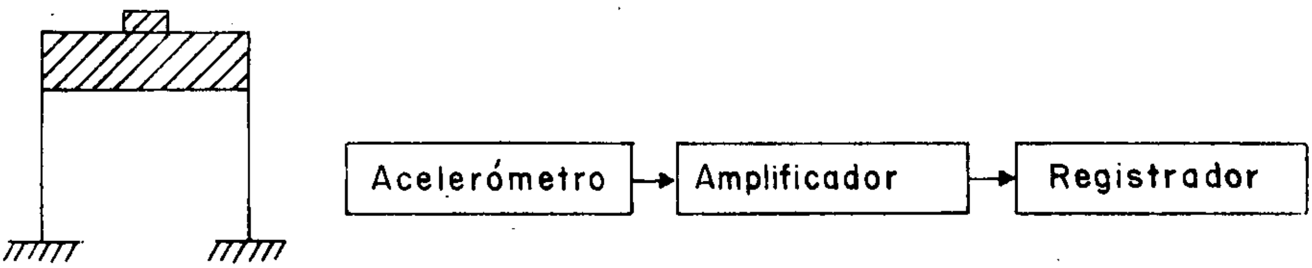
$$\boxed{L = \frac{2\pi\zeta}{\sqrt{1-\zeta^2}}} \qquad (10)$$

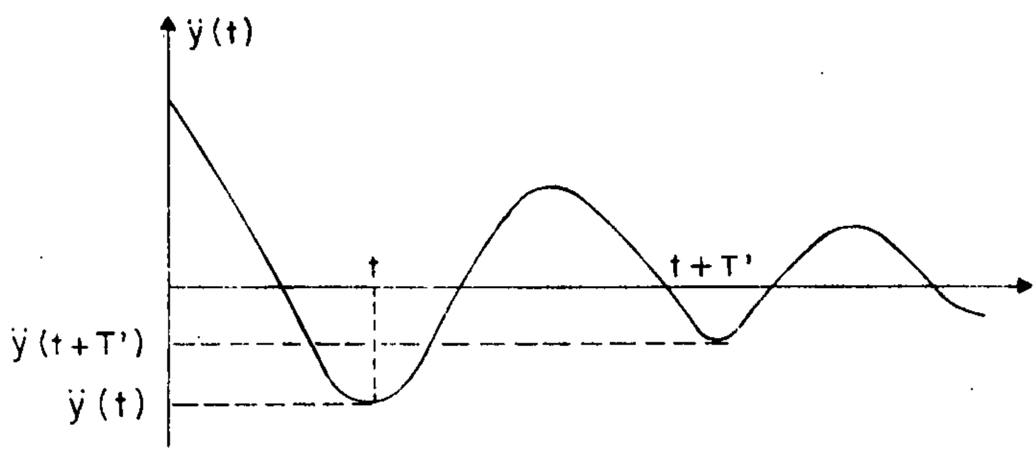
SI ζ ES PEQUEÑO,

$$\boxed{L \doteq 2\pi\zeta} \qquad (11)$$

DETERMINACION EXPERIMENTAL DE ζ EN ESTRUCTURAS REALES O EN MODELOS

SI SE REALIZA UN EXPERIMENTO EN EL CUAL SE SACA A LA ESTRUCTURA DE SU POSICION SE SACA A LA ESTRUCTURA DE SU POSICION DE EQUILIBRIO ESTATICO Y SE DEJA VIBRANDO LIBREMENTE, EL REGISTRO DE LAS ACELERACIONES QUE SE REGISTREN EN LA MASA TENDRA LA MISMA FORMA QUE LA GRAFICA DE LA EC. 7.



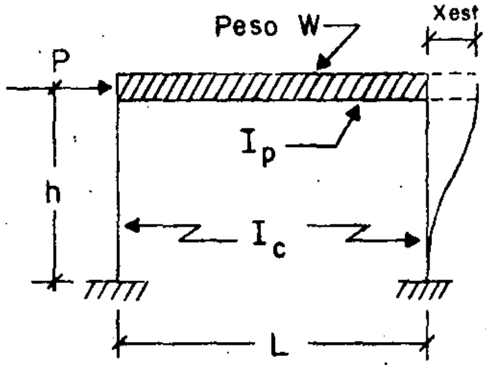


SI DE DICHO REGISTRO SE MIDEN $\ddot{y}(t + T')$ y $\ddot{y}(t)$ SE PUEDE OBTENER L Y, DE LA EC. (11), DESPEJAR A ζ

$$\zeta = \frac{L}{2\pi}$$

EJEMPLO

CALCULAR EL PERIODO NATURAL DE VIBRACION DE LA ESTRUCTURA MOSTRADA EN LA SIGUIENTE FIGURA:



$$K = \frac{P}{X_{est}}$$

P = carga estática

X_{est} = desplazamiento producido por P

I_c = momento de inercia de las columnas

I_p = momento de inercia del sistema de piso

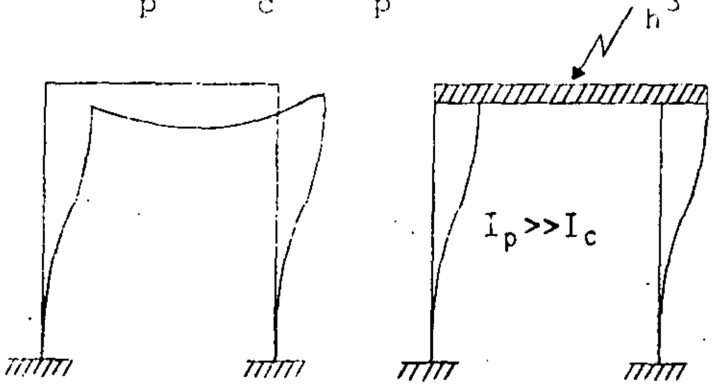
MEDIANTE EL ANALISIS ESTATICO DEL MARCO SE ENCUENTRA QUE

$$X_{est} = \frac{Ph^3}{6EI_c} \frac{\frac{3}{2} + \frac{I_c}{I_p}\frac{L}{h}}{6 + \frac{I_c}{I_p}\frac{L}{h}} \Rightarrow K = \frac{6EI}{h^3} \frac{6 + \frac{I_c}{I_p}\frac{L}{h}}{\frac{3}{2} + \frac{I_c}{I_p}\frac{L}{h}}$$

$$\text{Período natural} = T = \frac{2\pi}{\omega} = \frac{2\pi}{\sqrt{\frac{K}{m}}} = 2\pi\sqrt{\frac{W}{gK}}$$

$$T = 2\pi\sqrt{\frac{Wh^3}{g6EI} \frac{\frac{3}{2} + \frac{I_c}{I_p}\frac{L}{h}}{6 + \frac{I_c}{I_p}\frac{L}{h}}}\text{, en seg}$$

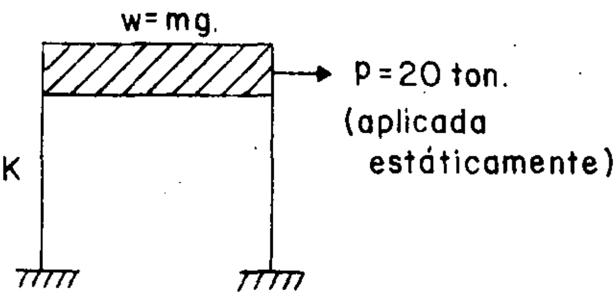
Si $I_p >> I_c$ $(I_p \to \infty)$, $K = \frac{24EI}{h^3}$

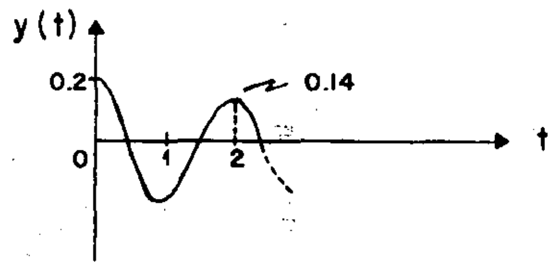


ESTRUCTURA DE CORTANTE: CUANDO LAS DEFORMACIONES OCURREN PRINCIPALMENTE DEBIDO A LA FUERZA CORTANTE DE ENTREPISO.

EJEMPLO

A UNA ESTRUCTURA DE UN PISO SE LE APLICA UNA CARGA HORIZONTAL DE 20 TON EN SU MASA, OBSERVANDOSE UN DESPLAZAMIENTO ESTATICO DE 0.2 CM. AL SOLTAR SUBITAMENTE LA FUERZA SE REGISTRA UN PERIODO DE OSCILACION DE 0.2 SEG, Y QUE LA AMPLITUD EN EL SEGUNDO CICLO ES DE 0.14 CM.





CALCULAR W, ω', f', L y ζ

1. DE $T' = \frac{2\pi}{\omega} = \frac{\pi 2}{\sqrt{\frac{K}{M}}} = \frac{2\pi\sqrt{W}}{\sqrt{Kg}} = 0.2$ Y $K = \frac{2.0}{0.2} = 100 \frac{TON}{CM}$

SE OBTIENE

$W = T'^2\ Kg/4\pi^2 = (0.2)^2 \times 100 \times 981/4\pi^2 = \frac{0.04 \times 100 \times 981}{4 \quad 9.87}$

$W = 99.4$ TON

2. $\omega' = \frac{2\pi}{T'} = \frac{2\pi}{0.2} = 10\pi\ \frac{RAD}{SEG}$; $f' = \frac{1}{T'} = \frac{1}{0.2} = 5$ cps

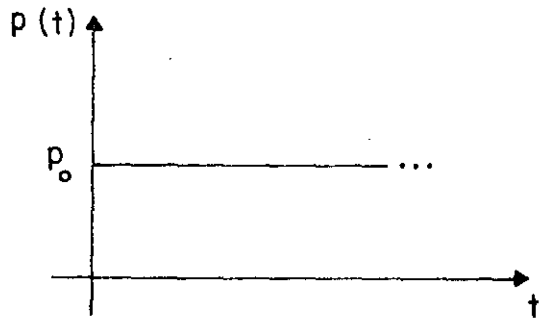
3. $L = \ln \frac{0.2}{0.14} = \ln 1.43 = 0.357$

$\zeta = \frac{L}{2\pi} = \frac{0.357}{2\pi} = 0.0568$ O $\zeta = 5.68\ \%$

$C = \zeta C_{cr} = \zeta 2\sqrt{KM} = 0.1132\sqrt{100 \times 99.4/981}$

EJEMPLO

CALCULAR LA RESPUESTA DE UN SISTEMA DE UN GRADO DE LIBERTAD SUJETO A LA SIGUIENTE EXCITACION, CON $C = 0$:



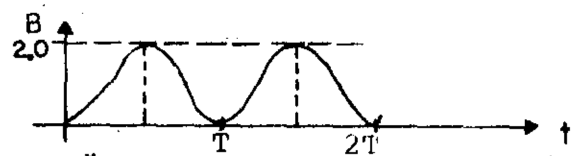
$$m\ddot{x} + kx = p_o$$

$$x = C_1 \operatorname{sen}\omega t + C_2 \cos\omega t + p_o/k$$

SI EN $t = 0$, $x = 0$ Y $\dot{x} = 0$:

$$C_2 = -p_o/k \quad \text{Y} \quad C_1 = 0$$

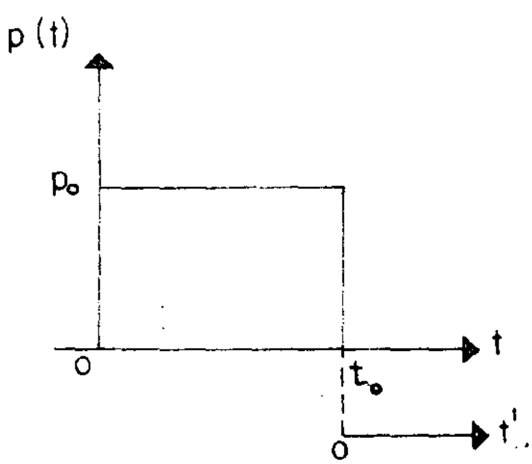
$$\therefore \quad x = \frac{p_o}{k}(1 - \cos\omega t);$$



$$B = \text{FACTOR DE AMPLIFICACION DINAMICA} = \frac{x}{\left(\frac{p_o}{k}\right)} = (1 - \cos\omega t)$$

$B_{MAX} = 2$, EN $t = T/2, 3T/2...$

AHORA, SI LA EXCITACION ES DE DURACION t_o



SI $t < t_o$:

$$x = \frac{p_o}{k}(1 - \cos\omega t)$$

$$\dot{x}(t) = \frac{\omega p_o}{k} \operatorname{sen}\omega t$$

EN $t = t_o$:

$$\left.\begin{aligned} x(t_o) &= \frac{p_o}{k}(1 - \cos\omega t_o) \\ \dot{x}(t_o) &= \frac{\omega p_o}{k} \operatorname{sen}\omega t_o \end{aligned}\right\} \text{CONDICIONES INICIALES PARA } t > t_o$$

SI $t>t_o$, $x = A\cos\omega t' + B\,\text{sen}\,\omega t'$, CON $t' = t - t_o$

EN $t' = 0$ ($t = t_o$), SE DEBEN CUMPLIR LAS CONDICIONES INICIALES ANTERIORES, LO CUAL CONDUCE A

$$A = \frac{p_o}{k}\,(1 - \cos\omega t_o) \qquad Y \qquad B = \frac{p_o}{k}\,\text{sen}\,\omega t_o$$

POR LO QUE $\quad x = \dfrac{p_o}{k}\,(1 - \cos\omega t_o)\,\cos\omega t' + \dfrac{p_o}{k}\,\text{sen}\,\omega t_o\,\text{sen}\,\omega t'$.

$$= \frac{p_o}{k}\,\sqrt{(1 - \cos\omega t_o)^2 + \text{sen}^2\,\omega t_o}\;\text{sen}(\omega t' - \theta)$$

O

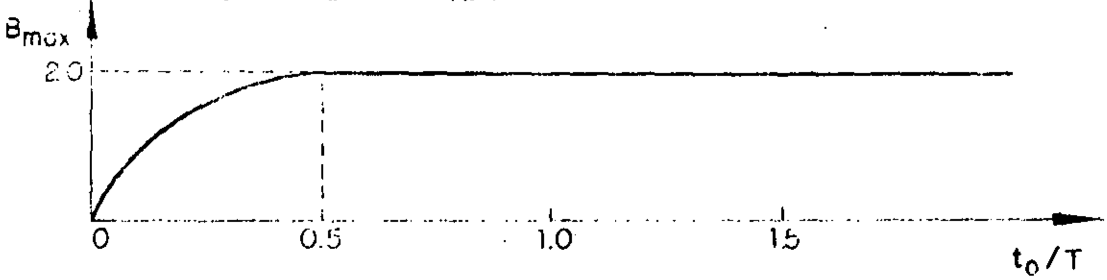
$$x = \frac{p_o}{k}\,\sqrt{2(1 - \cos\ t_o)}\;\;\text{sen}(\omega t' - \theta)$$

$$= \frac{p_o}{k}\;\underbrace{\left(2\,\text{sen}\frac{\omega t_o}{2}\right)\,\text{sen}(\omega t' - \theta)}$$

B= FACTOR DE AMPLIFICACION

$$B_{MAX} = 2\,\text{sen}\frac{\omega t_o}{2} = 2\,\text{sen}\left(\pi\frac{t_o}{T}\right)$$

CUANDO $\dfrac{\pi t_o}{T} = \dfrac{\pi}{2}$, $B_{MAX} = 2$



EL MAXIMO OCURRE DESPUES DE LA EXCITACION

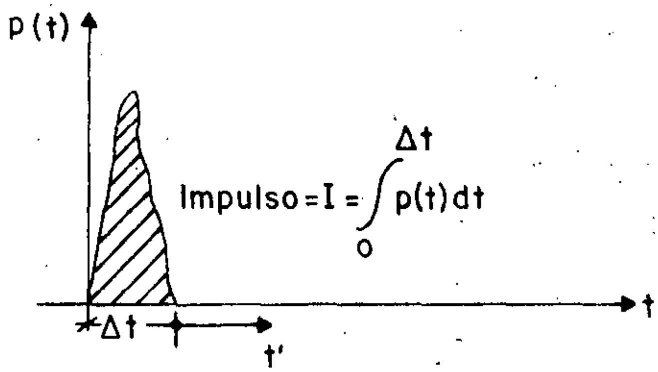
EL MAXIMO OCURRE DURANTE LA EXCITACION

SI t_o/T ES MUY PEQUEÑO, $\text{sen}\dfrac{\pi t_o}{T} = \pi t_o/T$

$$Y\ x_{MAX} = \frac{2p_o}{k}\frac{\pi t_o}{T} = \frac{2p_o}{m\frac{k}{m}}\frac{\omega t_o}{2} = \frac{p_o t_o}{m\omega} = \frac{I}{m\omega}$$

EN DONDE $I = p_o t_o$ = AREA BAJO LA EXCITACION

<u>EJEMPLO</u>: EXCITACION DADA POR UN IMPULSO.-SEA UN IMPULSO APLICADO DURANTE UN INTERVALO DE TIEMPO Δt MUY PEQUEÑO, TAL QUE $\Delta t/T << 1$:



POR EL PRINCIPIO IMPULSO - MOMENTO SE TIENE QUE

$$I = \int_0^{\Delta t} p(t)dt = m\dot{x} \qquad \Rightarrow \dot{x} = I/m$$

EN DONDE $\dot{x}$ ES LA VELOCIDAD QUE EL IMPULSO LE IMPRIME A LA MASA DEL SISTEMA. DESPUES DE Δt EL SISTEMA QUEDA VIBRANDO LIBREMENTE CON VELOCIDAD INICIAL $\dot{x}(0) = \frac{I}{m}$, MIDIENDO EL TIEMPO EN LA ESCALA DE t', Y CON DESPLAZAMIENTO INICIAL QUE PUEDE CONSIDERARSE NULO, DEBIDO A QUE EN EL CORTO INTERVALO DE TIEMPO Δt LA MASA ADQUIERE UN DESPLAZAMIENTO DE MAGNITUD DESPRECIABLE. EN TAL CASO LA RESPUESTA RESULTA

$$x(t') = \frac{\dot{x}(o)}{\omega}\ \text{sen}\,\omega t' = \frac{I}{m\omega}\text{sen}\,\omega t'$$

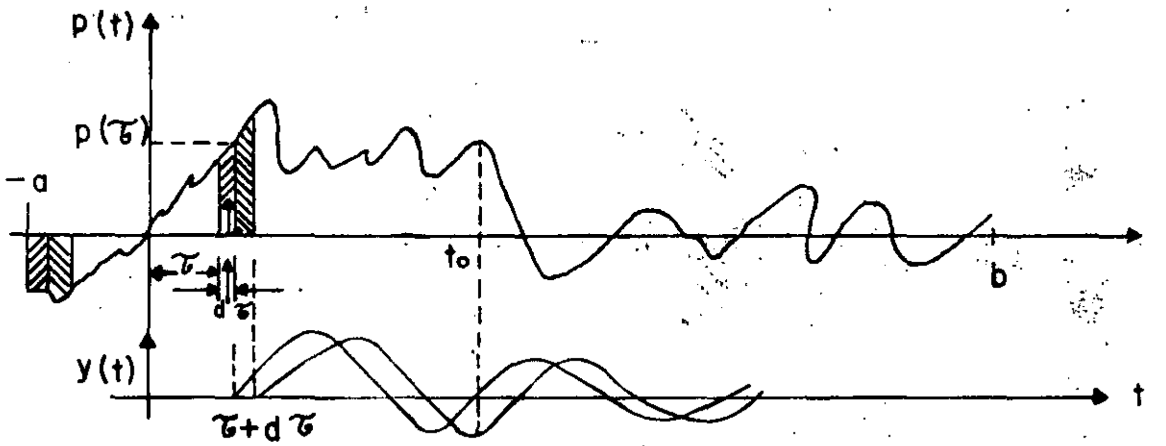
SI EL SISTEMA TIENE AMORTIGUAMIENTO,

$$x(t') = \frac{I}{m\omega}\ e^{-\zeta\omega t'}\ \text{sen}\,\omega' t'$$

SOLUCION AL PROBLEMA DE VIBRACIONES FORZADAS

A. FUERZA EXTERNA

VEAMOS PRIMERO EL CASO EN QUE EXISTE p(t) Y QUE $\ddot{x}_o(t) = 0$;
SIENDO p(t) ARBITRARIA



PUESTO QUE $d\tau \ll T$, LA FUERZA APLICADA EN $t=\tau$ PRODUCIRA UN INCREMENTO INSTANTANEO EN LA VELOCIDAD DE LA MASA IGUAL A

$$\dot{y} = \frac{p(\tau)d\tau}{M}$$

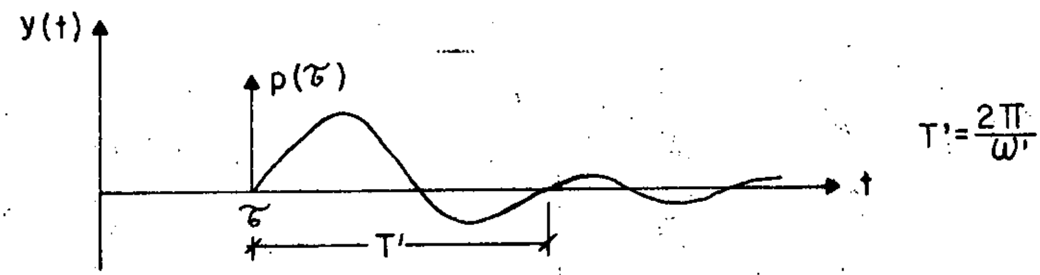
Y UN INCREMENTO INSTANTANEO NULO EN EL DESPLAZAMIENTO, ES DECIR, y=0. TOMANDO ESTOS INCREMENTOS COMO CONDICIONES INICIALES EN $t=\tau$, LA EC. 5 DA COMO RESULTADO

$$y(t) = \frac{p(\tau)d\tau}{M\omega'} \text{ sen } \omega'(t-\tau)\ e^{-h(t-\tau)} \quad ; \quad t \geq \tau$$

PUESTO QUE EL SISTEMA ES LINEAL ES POSIBLE SUPERPONER LOS EFECTOS OCASIONADOS POR LOS IMPULSOS APLICADOS EN CADA τ QUE HAYAN OCURRIDO ANTES DEL INSTANTE t DE INTERES; ES DECIR,

$$y(t) = \frac{1}{M\omega'} \int_{-\infty}^{t} p(\tau) e^{-h(t-\tau)} \operatorname{sen}\omega'(t-\tau)\, d\tau \qquad (12)$$

LA FUNCION $\frac{1}{M\omega'} e^{-h(t-\tau)} \operatorname{sen}\omega'(t-\tau)$, QUE ES LA RESPUESTA A UN IMPULSO INSTANTANEO UNITARIO DE FUERZA, SE LE CONOCE COMO <u>*FUNCION DE TRANSFERENCIA DEL SISTEMA*</u>.



LA SOLUCION DADA EN LA EC (12) SE DENOMINA INTEGRAL DE DUHAMEL. ESTA CONSTITUYE LA SOLUCION PARTICULAR DE LA ECUACION DIFERENCIAL DE EQUILIBRIO; LA SOLUCION GENERAL ES:

$$y(t) = Ae^{-ht} \cos(\omega' t - \theta) + \frac{1}{M\omega'} \int_{-\infty}^{t} p(\tau) e^{-h(t-\tau)} \operatorname{sen}\omega'(t-\tau)\, d\tau$$

EN DONDE A y θ DEPENDEN DE LAS CONDICIONES INICIALES DE DESPLAZAMIENTO Y VELOCIDAD, $y(0)$ Y $\dot{y}(0)$, RESPECTIVAMENTE. EN GENERAL LA PARTE DE LA RESPUESTA DADA POR LA SOLUCION PARTICULAR ES LA MAS IMPORTANTE, YA QUE LA OTRA PARTE SE AMORTIGUA RAPIDAMENTE.

B. <u>MOVIMIENTO DEL SUELO</u>

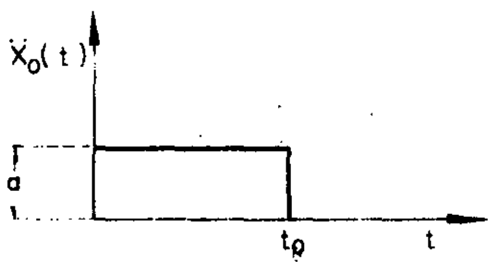
PARA ESCRIBIR LA SOLUCION PARTICULAR DE LA ECUACION DIFERENCIAL DE EQUILIBRIO PARA EL CASO DE VIBRACION FORZADA POR MOVIMIENTO DE LA BASE DE LA ESTRUCTURA, <u>BASTA CAMBIAR $p(\tau)/M$</u> DE LA EC (12) <u>POR $-\ddot{x}_o$</u>, YA QUE EN DICHA ECUACION APARECE EN EL MIEMBRO DERECHO $p(t)/M$ CUANDO LA EXCITACION ES $P(t)$ Y APARECE $-\ddot{x}_o$ CUANDO LA EXCITACION ES POR MOVIMIENTO DEL SUELO. EN ESTE CASO

LA SOLUCION PARTICULAR ES, ENTONCES.

$$\boxed{y(t) = \frac{-1}{\omega'} \int_{-\infty}^{t} \ddot{x}_o(\tau) e^{-h(t-\tau)} \operatorname{sen}\omega'(t-\tau)\, d\tau} \qquad (14)$$

EJEMPLO

CALCULAR LA RESPUESTA DE UN SISTEMA DE UN GRADO DE LIBERTAD CON AMORTIGUAMIENTO NULO, CUANDO LA EXCITACION ES LA SIGUIENTE:



$$\zeta = 0$$
$$\ddot{x}_o(t) = a, \text{ SI } 0 \leqslant t \leqslant t_o$$
$$\ddot{x}_o(t) = 0, \text{ SI } t<0 \text{ Ó } t>t_o$$

CONSIDERESE QUE $y(0)=0$ Y $\dot{y}(0)=0$. PUESTO QUE LAS CONDICIONES INICIALES SON NULAS SE TIENE QUE $A=0$ (UTILIZANDO LA EC (13) Y LA SOLUCION PARTICULAR QUE SIGUE, EC (A)):

$$y(t) = \frac{-1}{\omega} \int_{-\infty}^{t} a \operatorname{sen}\omega(t-\tau)\, d\tau = \frac{-a}{\omega} \int_{0}^{t} \operatorname{sen}\omega(t-\tau)\, d\tau$$

$$= \frac{-a}{\omega^2} (1-\cos\omega t) \qquad \text{SI} \quad 0 \leq t \leq t_o \qquad \text{(A)}$$

PARA FINES DE DISEÑO ESTRUCTURAL ES IMPORTANTE CONOCER LA RESPUESTA MAXIMA; ESTA OCURRE CUANDO $\cos\omega t = -1$, O SEA, CUANDO

$$\omega t = \pi \quad \text{Ó} \quad t = \frac{\pi}{\omega} = \frac{\pi}{\frac{2\pi}{T}} = \frac{T}{2}$$

Y VALE

$$\text{MAX}\ \{[y(t)]\} = \frac{2a}{\omega^2} = \frac{a}{2\pi^2} T^2, \text{ SI } 0 \leq \frac{T}{2} \leq t_o \text{ O } 0 \leq T \leq 2t_o$$

PARA $t>t_o$, O SEA, PARA $T/2>t_o$ ES NECESARIO OBTENER LA RESPUESTA EN VIBRACION LIBRE CON LAS CONDICIONES INICIALES DE VELOCIDAD Y DESPLAZAMIENTO CORRESPONDIENTES A $t=t_o$:

$$y(t_o) = \frac{-a}{\omega^2}(1 - \cos\omega t_o) \quad ; \quad \dot{y}(t_o) = \frac{-a}{\omega} \operatorname{sen}\omega t_o$$

APLICANDO LAS ECS. (5) Y (6) OBTENEMOS:

$$y(t) = \frac{-a}{\omega^2}[\operatorname{sen}\omega t_o \operatorname{sen}\omega t' - (1 - \cos\omega t_o)\cos\omega t']$$

$$= \frac{-a}{\omega^2}\sqrt{\operatorname{sen}^2\omega t_o + (1 - \cos\omega t_o)^2}\ \operatorname{sen}(\omega t' - \emptyset)$$

$$y(t) = \frac{-2a}{\omega^2} \operatorname{sen}\frac{\omega t_o}{2} \operatorname{sen}(\omega t' - \emptyset)$$

DONDE $t' = t - t_o$ Y $\emptyset = \tan^{-1}\left(\frac{1-\cos\omega t_o}{\operatorname{sen}\omega t_o}\right)$

EL VALOR MAXIMO DE LA RESPUESTA EN ESTE INTERVALO ES

$$\text{MAX}\{[y(t)]\} = \frac{2a}{\omega^2}\left|\operatorname{sen}\frac{\omega t_o}{2}\right| , \text{ SI } t>t_o \text{ O } T>2t_o$$

EXCITACION ARMONICA

CONSIDEREMOS AHORA EL CASO EN QUE LA ESTRUCTURA ES EXCITADA POR LA FUERZA ARMONICA

$$p(t) = p_o \operatorname{sen}\Omega t$$

DE DURACION INDEFINIDA.

LA SOLUCION DE ESTE PROBLEMA SE PUEDE ENCONTRAR SUSTITUYENDO A $p(t) = p_o \operatorname{sen}\Omega t$ EN LA INTEGRAL DE DUHAMEL Y OBTENIENDO SU SOLUCION. SIN EMBARGO, EL RESULTADO LO OBTENDREMOS DE LA CONSIDERACION DE QUE PARA QUE EL MIEMBRO DERECHO DE LA ECUACION DIFERENCIAL DE EQUILIBRIO APAREZCA UN TERMINO ARMONICO ES NECESARIO QUE EN EL IZQUIERDO SE TENGAN COMBINACIONES DE TERMINOS TAMBIEN ARMONICOS. CONSIDEREMOS, POR LO TANTO, LA SOLUCION

$$y(t) = A \operatorname{sen}\Omega t + B \cos\Omega t \qquad (14)$$

Y DETERMINEMOS LOS VALORES QUE DEBEN TENER A Y B PARA SATISFACER LA ECUACION DIFERENCIAL DE EQUILIBRIO, PARA LO CUAL HAY QUE SUSTITUIR A $y(t)$, $\dot{y}(t)$ Y $\ddot{y}(t)$ EN LA ECUACION DIFERENCIAL. HACIENDO ESTO Y FACTORIZANDO:

$$(-A\Omega^2 - 2h\Omega B + \omega^2 A) \operatorname{sen}\Omega t +$$

$$(-B\Omega^2 + 2hA\Omega + \omega^2 B) \cos\Omega t = \frac{p_o}{M} \operatorname{sen}\Omega t + 0 \times \cos\Omega t$$

PARA QUE ESTA IGUALDAD SE CUMPLA SE REQUIERE QUE

$$-A\Omega^2 - 2h\Omega B + \omega^2 A = \frac{p_o}{M}$$

$$-B\Omega^2 + 2h\Omega A + \omega^2 B = 0$$

RESOLVIENDO ESTE SISTEMA DE ECUACIONES SE OBTIENE:

$$A = \frac{\frac{P_o}{M}(\Omega^2 - \omega^2)}{(\omega^2 - \Omega^2)^2 + 4h^2\Omega^2}$$

$$B = \frac{-2h\Omega \frac{P_o}{M}}{(\omega^2 - \Omega^2)^2 + 4h^2\Omega^2}$$

SUSTITUYENDO A Y B EN LA EC. (14'):

$$y(t) = \frac{\frac{P_o}{M}}{(\omega^2 - \Omega^2)^2 + 4h^2\Omega^2}\{(\Omega^2-\omega^2)\ \mathrm{sen}\Omega t - 2h\Omega\ \cos\Omega t\} \quad (15)$$

O, TAMBIEN

$$y(t) = \frac{\frac{P_o}{M}}{\sqrt{(\omega^2 - \Omega^2)^2 + 4h^2\Omega^2}}\ \mathrm{sen}(\Omega t - \phi) \quad (16)$$

EN DONDE ϕ = ANG TAN $(\frac{-B}{A})$ = $\mathrm{TAN}^{-1} \frac{2h\Omega}{\omega^2 - \Omega^2}$ = ANGULO DE FASE (17)

DIVIDIENDO NUMERADOR Y DENOMINADOR DE LAS ECS (16) Y (17) ENTRE ω^2 SE OBTIENE:

$$\boxed{y(t) = \frac{\frac{P_o}{k}}{\sqrt{(1 - \frac{\Omega^2}{\omega^2})^2 + (2\zeta\frac{\Omega}{\omega})^2}}\ \mathrm{sen}(\Omega t - \phi)} \quad (18)$$

$$\boxed{\phi = \mathrm{TAN}^{-1} \frac{2\zeta\frac{\Omega}{\omega}}{1-\frac{\Omega^2}{\omega^2}}} \quad (19)$$

SOLUCION GENERAL PARA EL CASO $\xi = 0$

$$y(t) = C_1 \operatorname{sen} \omega t + C_2 \cos \omega t + \frac{P_o}{M} \cdot \frac{\operatorname{sen} \Omega t}{\omega^2 - \Omega^2}$$

SI EL SISTEMA PARTE DEL REPOSO, LAS CONDICIONES INICIALES SON $y(0) = 0$ y $\dot{y}(0) = 0$. EN ESTE CASO:

$$y(0) = 0 = C_1 \operatorname{sen} (\omega 0) + C_2 \cos (\omega 0) + \frac{P_o}{M} \quad \frac{\operatorname{sen} (\Omega 0)}{\omega^2 - \Omega^2} = 0$$

$$= 0 + C_2 + 0 = C_2 = 0$$

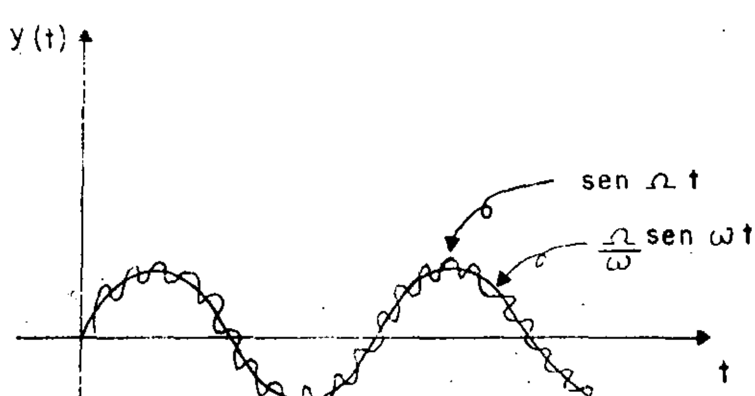
$$\dot{y}(0) = C_1 \omega \cos (\omega 0) - C_2 \omega \operatorname{sen} (\omega 0) + \frac{P_o \Omega}{M} \quad \frac{\cos (\Omega 0)}{\omega^2 - \Omega^2} = 0$$

$$= C_1 \omega + \frac{P_o \Omega}{M} \quad \frac{1}{\omega^2 - \Omega^2} = 0$$

$$\therefore \quad C_1 = \frac{-P_o}{M} \quad \frac{(\Omega / \omega)}{\omega^2 - \Omega^2}$$

$$y(t) = \frac{P_o}{M} \left(\frac{\operatorname{sen} \Omega t}{\omega^2 - \Omega^2} - \frac{\Omega}{\omega} \frac{\operatorname{sen} \omega t}{\omega^2 - \Omega^2} \right)$$

$$y(t) = \frac{(P_o / M)}{(1 - \Omega^2 / \omega^2)} \left[\operatorname{sen} \Omega t - \frac{\Omega}{\omega} \operatorname{sen} \omega t \right] \qquad (20')$$



SI SE TIENE EXCITACION ARMONICA EN LA BASE DE LA ESTRUCTURA $x_o(t) = a\,sen\Omega t$, O SEA, $\ddot{x}_o = -a\Omega^2 sen\Omega t$. BASTA CAMBIAR A p_o/M EN LA EC. (16) POR $-a\Omega^2$; HACIENDO ESTO SE OBTIENE

$$y(t) = \frac{(\Omega/\omega)^2}{\sqrt{\left(1-\frac{\Omega^2}{\omega^2}\right)^2 + \left(2\zeta\frac{\Omega}{\omega}\right)^2}}\, a\ sen(\Omega t - \emptyset) \tag{20}$$

FACTOR DE AMPLIFICACION DINAMICA DE DESPL. = $B_d = MAX\left[\frac{y(t)}{a}\right]$

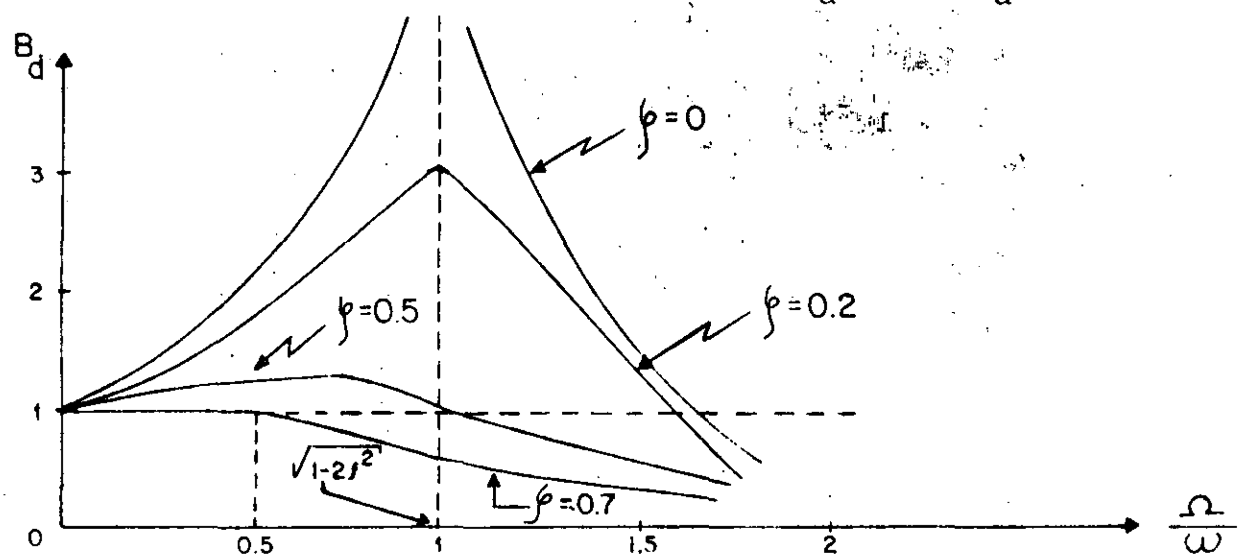


FIG. 1. CURVAS DE AMPLIFICACION DINAMICA PARA EL CASO DE FUERZA EXTERNA

$$\boxed{B_d = \frac{1}{\sqrt{\left(1-\frac{\Omega^2}{\omega^2}\right)^2 + \left(2\zeta\frac{\Omega}{\omega}\right)^2}}} \tag{21}$$

LOS FACTORES DE AMPLIFICACION DINAMICA DE VELOCIDAD Y ACELERACION SE SE PUEDEN OBTENER DERIVANDO RESPECTO A t LA EC. (16) O LA (20), SEGUN SEA EL CASO. LOS RESULTADOS SON, RESPECTIVAMENTE,

$$MAX\left[\frac{\dot{y}(t)}{a\omega}\right] = B_v = \frac{\Omega}{\omega} B_d \quad Y \quad B_a = \left(\frac{\Omega}{\omega}\right)^2 B_d = MAX\left[\frac{\ddot{y}(t)}{a\omega^2}\right] \tag{22}$$

EJEMPLO

CON UNA MAQUINA VIBRATORIA PORTATIL QUE PRODUCE FUERZAS ARMONICAS SE PROBO UNA ESTRUCTURA, AJUSTANDO LA MAQUINA EN LAS FRECUENCIAS $\Omega_1 = 16\ \frac{RAD}{SEG}$ Y $\Omega_2 = 25\ \frac{RAD}{SEG}$, CON UNA FUERZA MAXIMA DE 500 LB EN CADA CASO. LAS AMPLITUDES Y ANGULOS DE FASE DE LA RESPUESTA QUE SE MIDIERON FUERON:

$$\rho_1 = 7.2 \times 10^{-3} in,\ \emptyset_1 = 15° \ (\cos\emptyset_1 = 0.966\ ;\ sen\emptyset_1 = 0.259)$$

$$\rho_2 = 14.5 \times 10^{-3} in,\ \emptyset_2 = 55° (\cos\emptyset_2 = 0.574;\ sen\emptyset_2 = 0.819)$$

EVALUAR LAS PROPIEDADES DINAMICAS DEL SISTEMA.

HACIENDO:

$$\rho_i = \frac{p_o}{k} B_{d_i} = \frac{p_o}{k} \frac{1}{1-\beta^2} \underbrace{\left\{\frac{1}{1 + |2\zeta\beta/(1-\beta^2)|^2}\right\}_i^{1/2}}_{\cos\emptyset_i}$$

$$\rho_i = \frac{p_o}{k} \frac{\cos\emptyset_i}{1-\beta^2} \quad ; \quad \beta = \Omega/\omega$$

O

$$k - k\beta^2 = \frac{p_o \cos\emptyset_i}{\rho_i} = k - \Omega^2 m \tag{23}$$

SUSTITUYENDO LOS VALORES EXPERIMENTALES DE LAS DOS PRUEBAS:

$$\left.\begin{aligned} k - (16)^2 m &= \frac{500\ (0.966)}{7.2 \times 10^3} \\ k - (25)^2 m &= \frac{500\ (0.574)}{14.5 \times 10^{-3}} \end{aligned}\right\} \qquad \begin{aligned} k &= 100\ 000\ \frac{lb}{in} \\ m &= 128.5\ \frac{lb\ SEG^2}{in} \end{aligned}$$

$$\Downarrow$$

$$\omega = \sqrt{\frac{k}{m}} = 27.9\ \frac{RAD}{SEG}$$

USANDO LAS ECS. (17) Y (23) SE OBTIENE:

$$\zeta = \frac{p_o \,\mathrm{sen}\,\emptyset_i}{2\beta_i \; k\rho_i} \;;\; \text{DE DONDE } \zeta \; \frac{500\,(0.259)}{2\frac{16}{27.9}\,100\,000(7.2 \times 10^{-3})} = 15.7\%$$

RESONANCIA

CUANDO LA EXCITACION TIENE FRECUENCIA IGUAL A LA NATURAL DEL SISTEMA, SE DICE QUE SE PRESENTA EL CASO DE RESONANCIA. DE LA EC.(20) ES EVIDENTE QUE SI $\beta=\Omega/\omega=1$ SE TIENE

$$y(t) = \underbrace{\frac{1}{2\zeta}}_{B_d} a \;\mathrm{sen}(\Omega t-\emptyset)$$

O $(B_d)_{res} = \frac{1}{2\zeta}$ EN CASO DE MOVIMIENTO DEL SUELO Y DE FUERZA EXTERNA

SIN EMBARGO, AUNQUE ESTA RESPUESTA ES CASI IGUAL A LA MAXIMA, ESTA OCURRE CUANDO $\Omega = \omega\sqrt{1-2\zeta^2}$. EN EL CASO DE $\dot{y}(t)$ Y $\ddot{y}(t)$, EL MAXIMO OCURRE, RESPECTIVAMENTE, CUANDO

$\Omega = \omega$ Y $\Omega = \frac{\omega}{\sqrt{1-2\zeta^2}}$ SI $\zeta \leq 20\%$, LOS VALORES DE ESTAS Ω NO DIFIEREN EN MAS DE 2%.

EL MAXIMO VALOR DE B_d (PARA $\Omega = \omega\sqrt{1-2\zeta^2}$) ES

$$(B_d)_{MAX} = \frac{1}{2\zeta\sqrt{1-\zeta^2}} \qquad \text{O} \qquad (B_d)_{MAX} = \frac{(\Omega/\omega)^2}{2\zeta\sqrt{1-\zeta^2}}$$

SI SE TIENE FUERZA EXTERNA O MOVIMIENTO DEL SUELO, RESPECTIVAMENTE.

SE OBSERVA EN ESTAS ECUACIONES QUE SI $\zeta=0$, $(B_d)_{MAX} = \infty$.

SI SE ANALIZA LA SOLUCION GENERAL DE LA ECUACION DIFERENCIAL DE MOVIMIENTO PARA EL CASO DE CONDICIONES INICIALES NULAS Y $\beta=1$ SE TIENE QUE:

$$y(t) = e^{-ht}(A \operatorname{sen} \omega' t + B \cos \omega' t) - \frac{p_o}{k} \frac{\cos \omega t}{2\zeta}$$

$$y(0) = B - p_o/(2\zeta k) = 0$$

DE DONDE, HACIENDO $y(0)=0$ Y $\dot{y}(0)=0$, SE OBTIENEN:

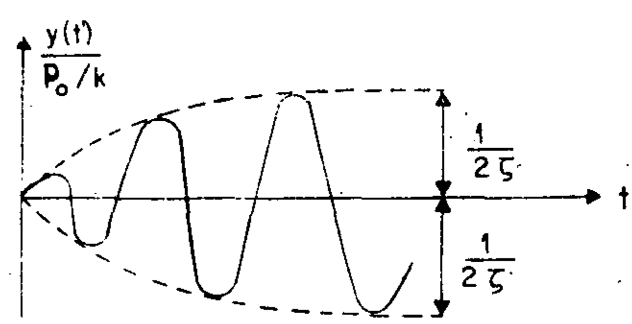
$$A = \frac{p_o}{k} \frac{\omega}{2\omega'} = \frac{p_o}{k} \frac{1}{2\sqrt{1-\zeta^2}} \quad ; \quad B = \frac{p_o}{k} \frac{1}{2\zeta}$$

POR LO QUE

$$y(t) = \frac{1}{2\zeta} \frac{p_o}{k} \left[e^{-ht} \left(\frac{\zeta}{\sqrt{1-\zeta^2}} \operatorname{sen}\omega' t + \cos\omega' t\right) - \cos\omega t\right]$$

PARA AMORTIGUAMIENTOS PEQUEÑOS:

$$\frac{y(t)}{p_o/k} = \frac{1}{2\zeta} (e^{-ht}-1)\cos\omega t$$

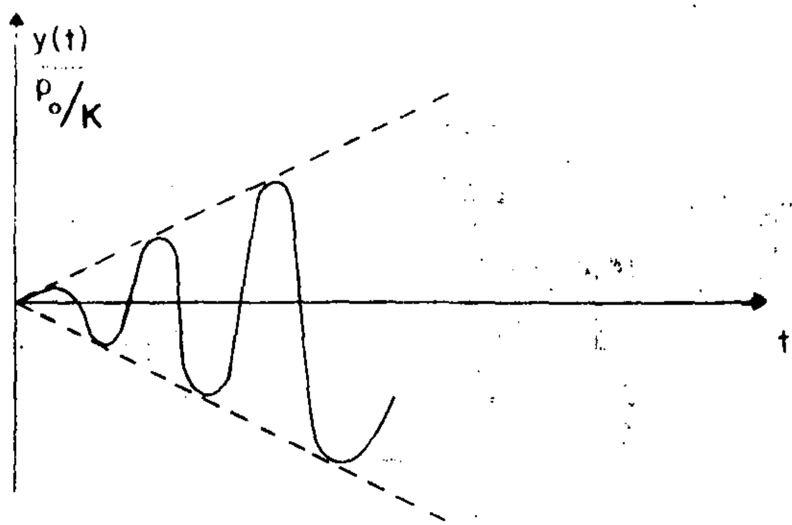


Si $\zeta>0$ y $\beta=1$

SI $\zeta=0$, APLICANDO LA REGLA DE L'HOSPITAL, SE OBTIENE:

$$\frac{y(t)}{p_o/k} = \frac{1}{2} (\operatorname{sen}\omega t - \omega t \cos\omega t)$$

O SEA, EL MAXIMO DE LA RESPUESTA TIENDE A INFINITO GRADUALMENTE.



CARACTERISTICAS DINAMICAS DE LOS REGISTRADORES DE SISMOS.

SI LA ACELERACION DE LA BASE DE UN INSTRUMENTO ES ARMONICA, DADA POR LA ECUACION

$$\ddot{x}_o(t) = a \operatorname{sen}\Omega t$$

EL FACTOR DE AMPLIFICACION RESULTA SER

$$\bar{B}_d = \frac{1}{\sqrt{\left(1-\frac{\Omega^2}{\omega^2}\right)^2 + \left(2\zeta\frac{\Omega}{\omega}\right)^2}} \quad \frac{1}{\omega^2} = \frac{B_d}{\omega^2}$$

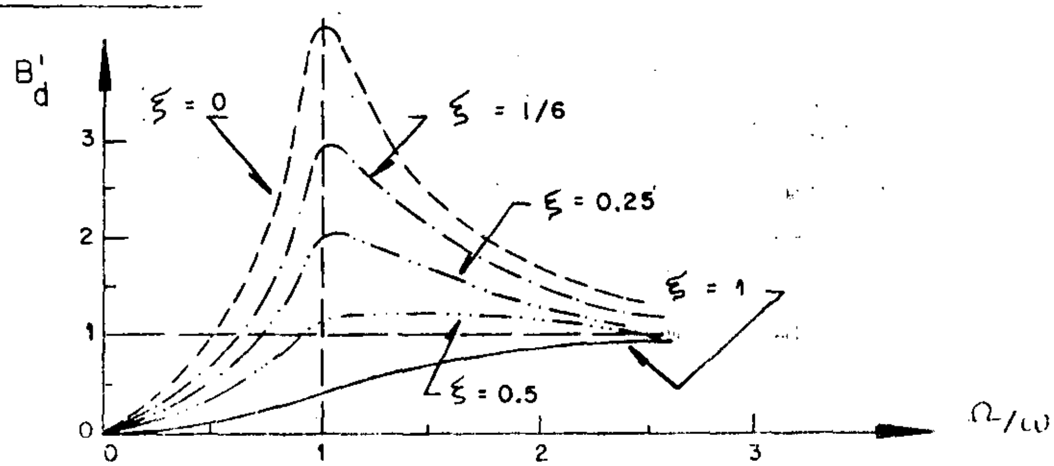
PUESTO QUE LA FIG I CORRESPONDE A B_d, Y EN ELLA SE OBSERVA QUE PARA $\zeta = 0.7$ SE TIENE $B_d = 1$ PARA $0 \leq \Omega/\omega \leq 0.6$, SE CONCLUYE QUE EL DESPLAZAMIENTO DE LA MASA DE UN SISTEMA ES PROPORCIONAL A LA ACELERACION DE SU BASE, SI ESTE TIENE AMORTIGUAMIENTO DEL 70% Y SI LAS EXCITACIONES QUE SE TRATAN DE REGISTRAR TIENEN FRECUENCIAS INFERIORES AL 60% DE LA FRECUENCIA NATURAL DEL SISTEMA. SI ESTO SE CUMPLE, EL APARATO RESULTA SER UN ACELEROMETRO.

EN INGENIERIA SISMICA LA MAXIMA FRECUENCIA DE INTERES ES DEL ORDEN DE 10 CPS (T = 0.1 SEG), POR LO QUE LOS ACELEROMETROS TIENEN FRECUENCIA NATURAL DE 16 A 20 CPS.

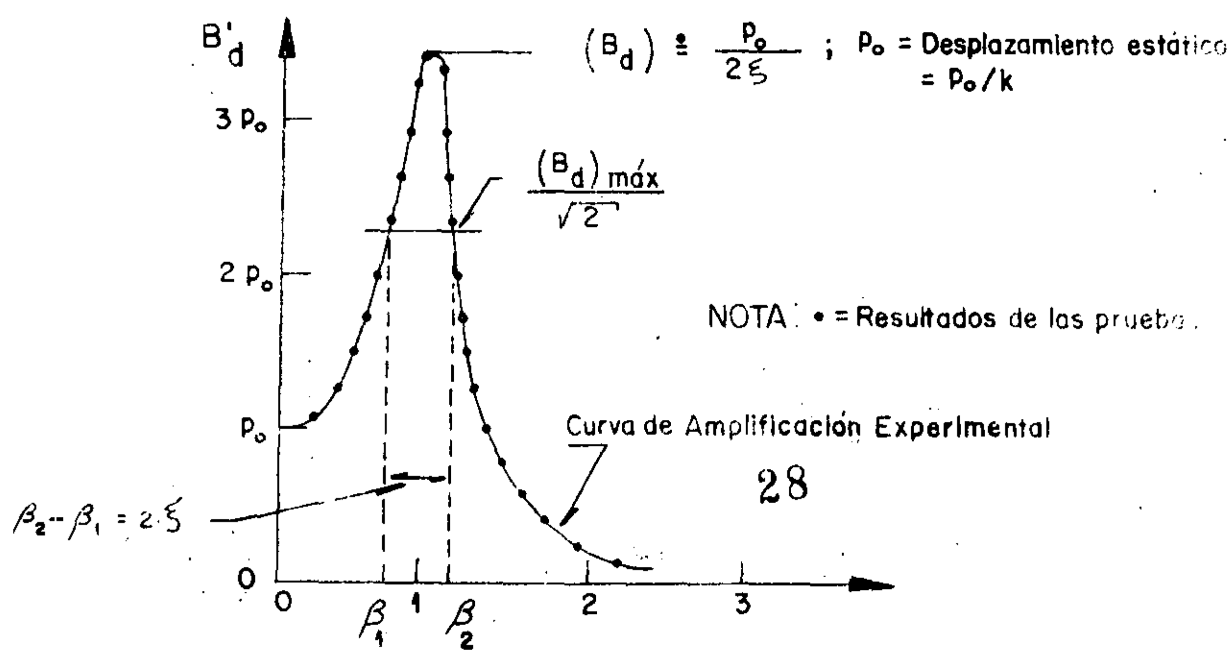
POR OTRA PARTE SI LA EXCITACION DEL SUELO ES $x_o = a\ \text{sen}\Omega t$, O SEA, $\ddot{x} = -a\ \Omega^2 \text{sen}\Omega t$, ENTONCES EL FACTOR DE AMPLIFICACION RESULTA SER EL SEÑALADO EN LA ECUACION (20), ES DECIR,

$$B'_d = \frac{(\Omega/\omega)^2}{\sqrt{(1-(\Omega/\omega)^2) + (2\zeta\Omega/\omega)^2}}$$

EN LA GRAFICA CORRESPONDIENTE SE OBSERVA QUE SI $\zeta=0.5$ Y $\Omega>\omega$ EL DESPLAZAMIENTO DE LA MASA ES PROPORCIONAL AL DEL SUELO; SI ESTO SE CUMPLE, EL APARATO, CONSTITUYE UN DESPLAZOMETRO, CONOCIDO TAMBIEN COMO SISMOMETRO.



DETERMINACION EXPERIMENTAL DEL AMORTIGUAMIENTO DE UNA ESTRUCTURA MEDIANTE VIBRACIONES FORZADAS ARMONICAS

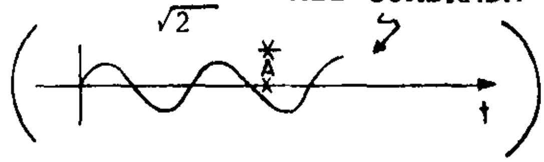


SI SE DETERMINA B_d EXPERIMENTALMENTE MEDIANTE UNA SERIE DE PRUEBAS DE VIBRACION FORZADA CON FUERZAS ARMONICAS, Y ADEMAS SE DETERMINA ρ_o, ENTONCES

$$\boxed{\zeta \doteq \frac{\rho_o}{2(B_d)_{MAX}}} \qquad (24)$$

OTRO METODO PARA DETERMINAR ζ CON BASE EN LA CURVA EXPERIMENTAL DE B_d SE CONOCE CON EL NOMBRE DE "<u>METODO DEL ANCHO DE BANDA DE LA MITAD DE POTENCIA</u>". ESTE SE BASA EN DETERMINAR LAS FRECUENCIAS QUE CORRESPONDEN AL VALOR rms DE LA AMPLITUD EN RESONANCIA, EL CUAL VALE $(B_d)_{MAX}/\sqrt{2}$; SEAN β_2 Y β_1 ESTAS FRECUENCIAS. DE LA ECUACION DE B_d SE OBTIENE:

$\text{rms} = \frac{A}{\sqrt{2}}$ = RAIZ CUADRADA DEL VALOR MEDIO CUADRATICO



$$\frac{1}{\sqrt{2}} \; \frac{\rho_o}{2\zeta} = \rho_o / \sqrt{(1-\beta^2)^2 + (2\zeta\beta)^2}$$

ELEVANDO AL CUADRADO AMBOS MIEMBROS:

$$\frac{1}{8\zeta^2} = \frac{1}{(1-\beta^2)^2 + (2\zeta\beta)^2}$$

DE DONDE $\beta^2 = 1 - 2\zeta^2 \pm 2\zeta\sqrt{1+\zeta^2}$

DE AQUI, DESPRECIANDO EL TERMINO ζ^2 DEL RADICAL, SE OBTIENE

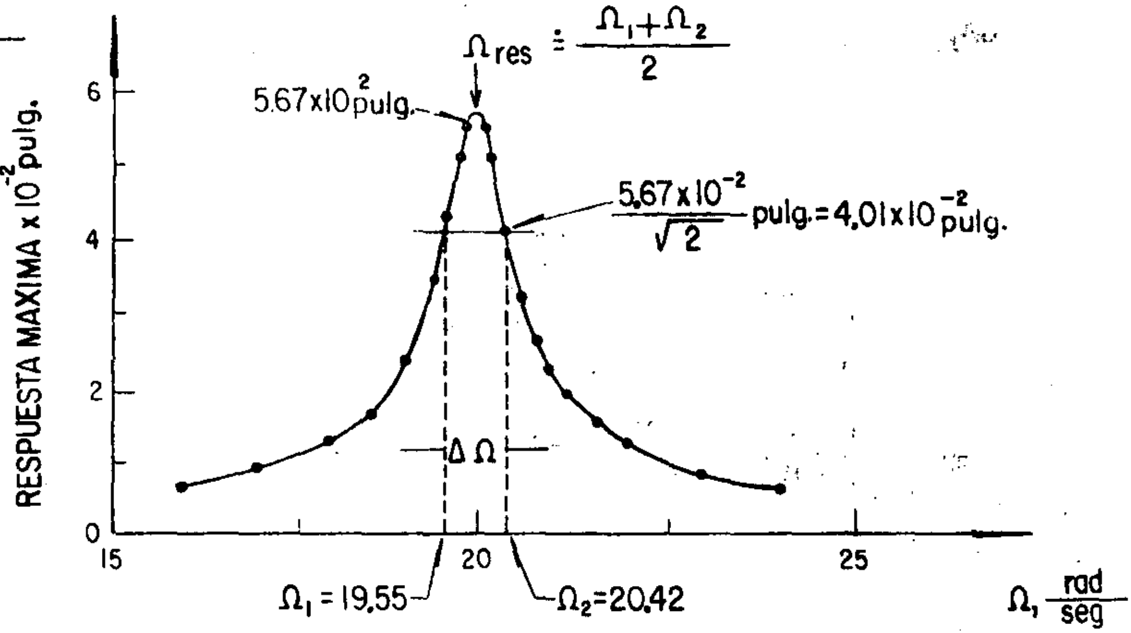
$$\beta_1^2 \doteq 1 - 2\zeta - 2\zeta^2 \quad ; \quad \beta_1 \doteq 1 - \zeta - \zeta^2$$

$$\beta_2^2 \doteq 1 + 2\zeta - 2\zeta^2 \quad ; \quad \beta_2 \doteq 1 + \zeta - \zeta^2$$

$$\beta_2 - \beta_1 \doteq 2\zeta$$

DE DONDE $$\boxed{\zeta \doteq \frac{\beta_2 - \beta_1}{2}} \qquad (25)$$

EJEMPLO



DE LA EC (25) $\qquad \Delta\Omega = \Omega_2 - \Omega_1 = 0.87 \ \frac{RAD}{SEG}$

$$\therefore \ \doteq \ \frac{\beta_2 - \beta_1}{2} = \frac{\frac{\Omega_2}{\Omega_{res}} - \frac{\Omega_1}{\Omega_{res}}}{2} = \frac{\Omega_2 - \Omega_1}{\Omega_2 + \Omega_1} = \frac{0.87}{39.97} = 2.18\%$$

METODO NUMERICO β DE NEWMARK PARA RESOLVER EL PROBLEMA DE VIBRACIONES FORZADAS.

EL METODO QUE A CONTINUACION SE DESCRIBE ES ADAPTABLE A SISTEMAS NO LINEALES CON VARIOS GRADOS DE LIBERTAD.

PROCEDIMIENTO:

1. SEAN y_i, $\dot{y}_i$ y $\ddot{y}_i$, CONOCIDOS EN EL INSTANTE t_i, Y $t_{i+1} = t_i + \Delta t$. SUPONGAMOS EL VALOR DE $\ddot{y}_{i+1}$

2. CALCULEMOS $\dot{y}_{i+1} \doteq \dot{y}_i + (\ddot{y}_i + \ddot{y}_{i+1})\Delta t/2 \qquad (26)$

3. CALCULEMOS $y_{i+1} \doteq y_i + \dot{y}_i \Delta t + (\frac{1}{2} - \beta)\ddot{y}_i(\Delta t)^2 + \beta\ddot{y}_{i+1}(\Delta t)$ (27)

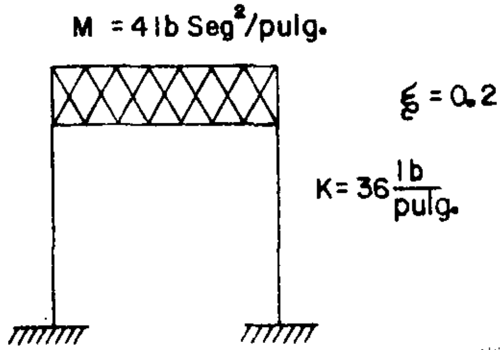
4. CALCULEMOS UNA NUEVA APROXIMACION PARA $\ddot{y}_{i+1}$ A PARTIR DE LA ECUACION DIFERENCIAL DE EQUILIBRIO:

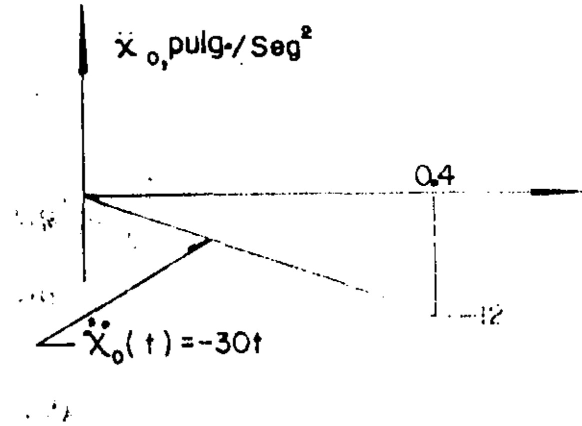
$$\ddot{y}_{i+1} \doteq -2\zeta\omega\dot{y}_{i+1} - \omega^2(y_{i+1} - y_{est}) - (\ddot{x}_o)_{i+1} \quad (28)$$

DONDE $y_{est} = p(t_{i+1})/k$

5. REPITAMOS LAS ETAPAS 2 A 4 EMPEZANDO CON EL NUEVO VALOR $\ddot{y}_{i+1}$ HASTA QUE EN DOS CICLOS CONSECUTIVOS SE TENGAN VALORES DE $\ddot{y}_{i+1}$ CASI IGUALES.

SE RECOMIENDAN VALORES DE <u>β DE 1/6 A 1/4 Y $\Delta t \doteq 0.1T$</u> PARA ASEGURAR CONVERGENCIA Y ESTABILIDAD.





CALCULAR LA RESPUESTA DE LA ESTRUCTURA APLICANDO EL METODO β DE NEWMARK

$$\omega = \sqrt{K/M} = \sqrt{36/4} = 3\ \frac{RAD}{SEG}$$

$$h = \xi\omega = 0.2 \times 3 = 0.6 \quad ; \quad T = \frac{2\pi}{3} = 2.09\ SEG$$

TOMAREMOS β=0.2 Y Δt = 0.2 ($\doteq$ o.1T) SUSTITUYENDO EN LAS ECS (26), (27) y (28):

$$\dot{y}_{i+1} \doteq \dot{y}_i + 0.1\ (\ddot{y}_i + \ddot{y}_{i+1})$$

$$y_{i+1} \doteq y_i + 0.2\dot{y}_i + 0.012\ddot{y}_i + 0.008\ddot{y}_{i+1}$$

$$\ddot{y}_{i+1} = -1.2\dot{y}_{i+1} - 9y_{i+1} - (\ddot{x}_o)_{i+1}$$

EN t=0 SABEMOS QUE SE TIENE y=0, $\dot{y}$=0 Y $\ddot{y}$=0

EN t=0 + Δt = 0.2 SEG; SUPONGAMOS $\ddot{y}_{i+1}$ = 5.0 IN/SEG^2; $\ddot{x}_o$ = -6

y_i = 0

$\dot{y}_i$ = 0

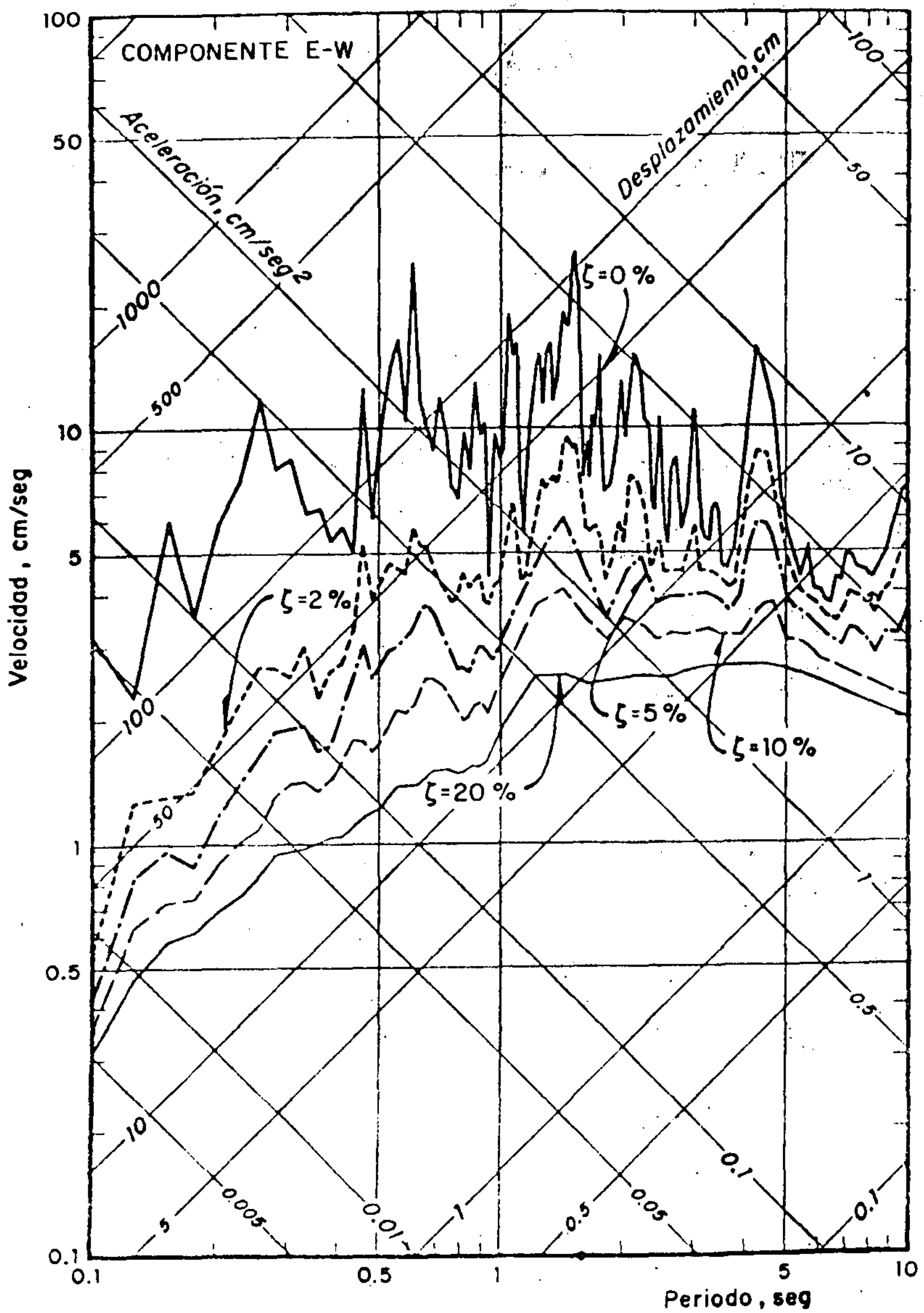
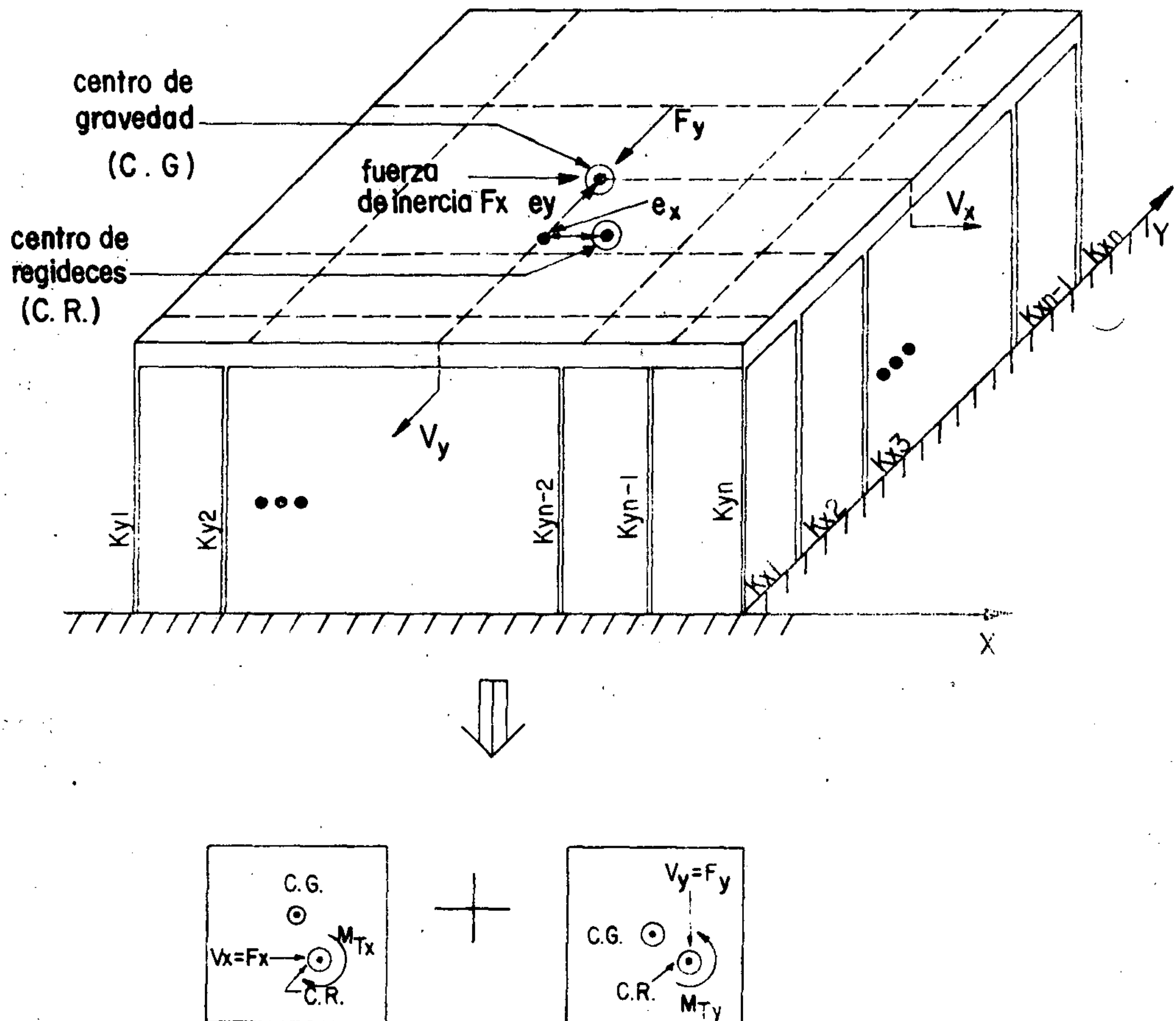
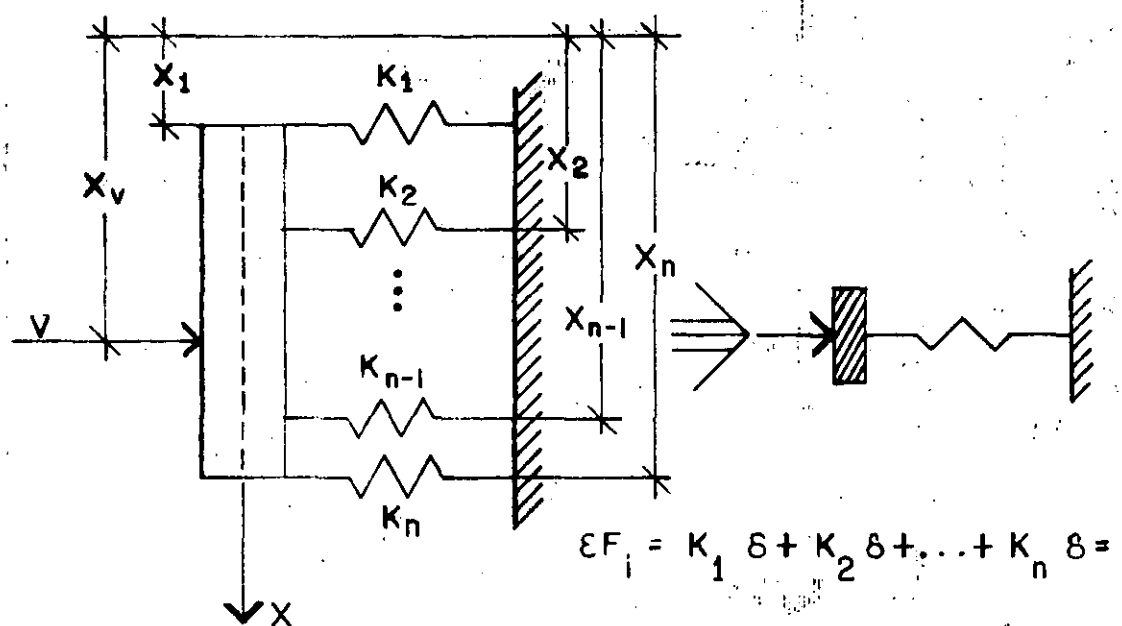


Fig 116 Espectros de respuesta. Ciudad Universitaria, 6 de julio de 1964

DISTRIBUCION DE FUERZAS CORTANTES DIRECTAS Y POR TORSION





$$\Sigma F_i = K_1\,\delta + K_2\,\delta + \ldots + K_n\,\delta = V = K_{eq}\,\delta$$

$$\delta \Sigma K_i = \delta\, K_{eq}$$

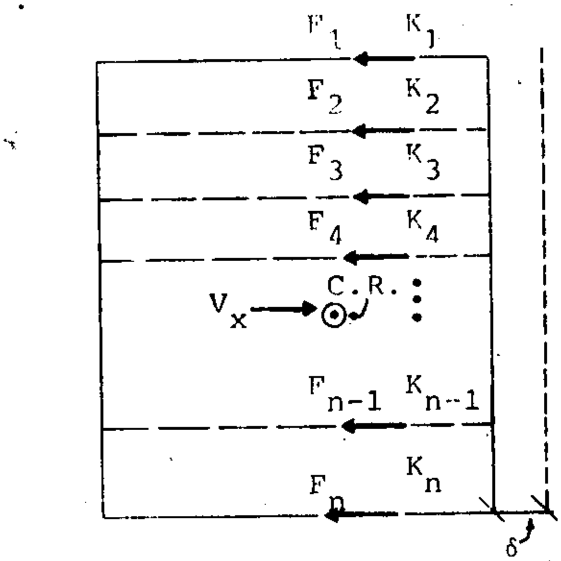
$$\boxed{K_{eq} = \sum_{i=1}^{n} K_i}$$

$$\Sigma M_i = \Sigma F_i X_i = \Sigma K_i \delta X_i = \delta K_i X_i = VX = K_{eq}\,\delta X_v$$

$$\boxed{X_v = \frac{\sum_{i=i}^{n} K_i X_i}{\sum_{i=i}^{n} K_i}}$$

⟸ POSICION DEL CENTRO DE RIGIDECES

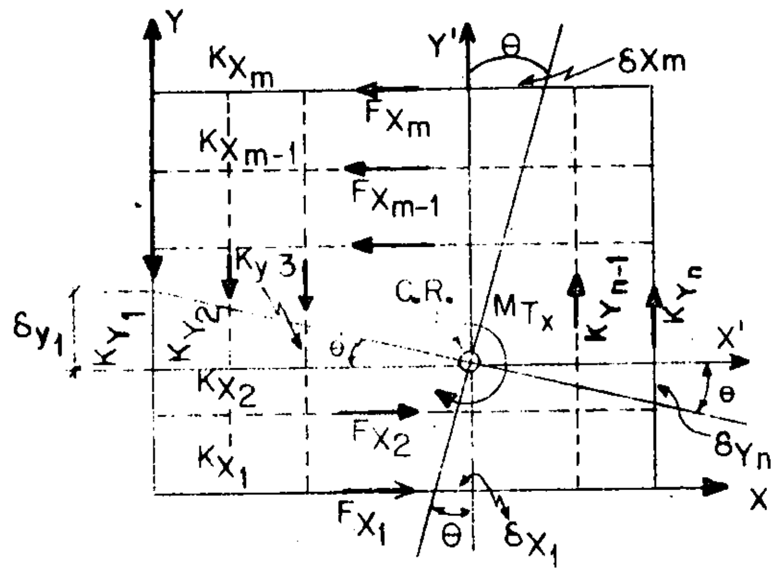
VEAMOS COMO SE DISTRIBUYEN LAS FUERZAS CORTANTES EN LOS MARCOS



$$F_i = K_i \delta$$

$$\Sigma F_i = \Sigma K_i \delta = V_x \therefore \delta = \frac{V_x}{\Sigma K_i}$$

$$\boxed{F_i = V_x \frac{K_i}{\sum_{i=1}^{n} K_i}}$$



$$F_{x_i} = K_{x_i}\delta_{x_i} = K_{x_i} Y'_i\,\theta$$

$$F_{x_i} = K_{y_i}\delta_{y_i} = K_{y_i} X'_i\,\theta$$

$$\Sigma M_{C.R.} = \Sigma F_{x_i} Y'_i + \Sigma F_{y_i} X'_i$$

$$= \theta(\Sigma K_{x_i} Y_i'^2 + \Sigma K_{y_i} X_i'^2)$$

$$= M_{T_x}$$

DE DONDE $\theta = \dfrac{M_T}{\Sigma K_{x_i} Y_i'^2 + K_{y_i} X_i'^2}$

POR LO QUE

$$\boxed{F_{x_i} = M_{TX} \frac{K_{x_i} Y'_i}{\Sigma K_{x_i} Y_i'^2 + K_{y_i} X_i'^2}} \quad ; \quad \boxed{F_{y_i} = M_{TX} \frac{K_{y_i} X'_i}{\Sigma K_{x_i} Y_i'^2 + \Sigma K_{y_i} X_i'^2}}$$

SISTEMAS NO LINEALES DE UN GRADO DE LIBERTAD

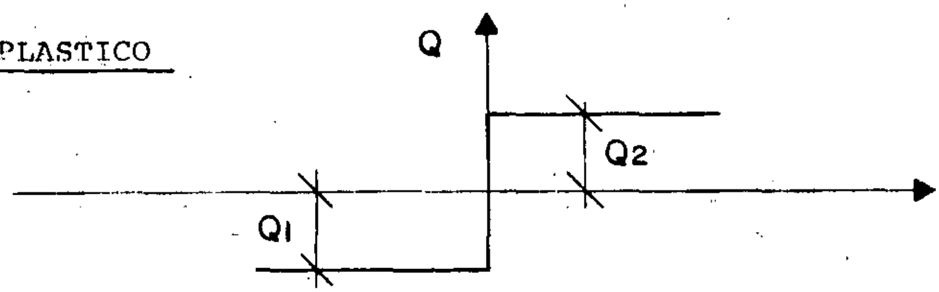
ECUACION DE MOVIMIENTO:

$$M\ddot{x} + Q(y,\dot{y}) = P(t) \quad ; \quad y = x-x_o = \text{DESPLAZAMIENTO RELATIVO}$$

SI $Q(y,\dot{y}) = KY + C\dot{Y}$ SE TIENE EL SISTEMA ELASTICO LINEAL

MODELOS PARTICULARES

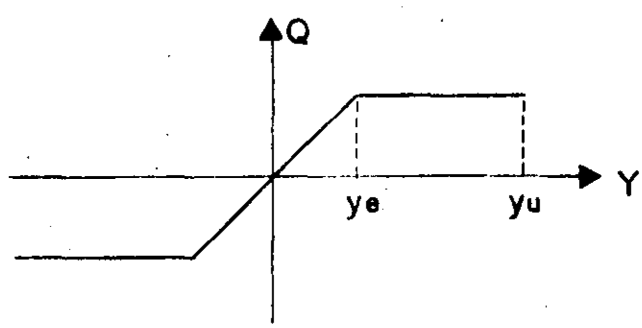
1. <u>RIGIDO-PLASTICO</u>



$Q = -Q_1 + C\dot{y}$, SI $\dot{y}<0$

$Q = Q_2 + C\dot{y}$, SI $\dot{y}<0$ EN DONDE C = CONSTANTE SE HA EMPLEADO COMO MODELO EN EL ANALISIS DE TALUDES Y CORTINAS DE PRESAS DE TIERRA Y ENROCAMIENTO

2. <u>ELASTO-PLASTICO</u>



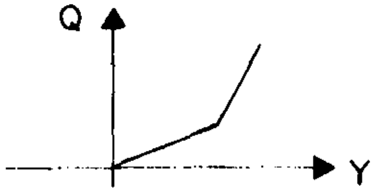
$Q = Q_1(y) + C\dot{y}$

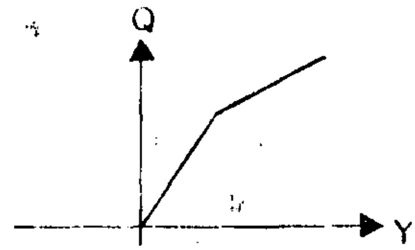
SE EMPLEA COMO MODELO EN EL ANALISIS DE ESTRUCTURAS DUCTILES.

<u>FACTOR DE DUCTILIDAD</u> $= \mu = y_u/y_e$

y_u = DESPLAZAMIENTO MAXIMO QUE PUEDE SOPORTAR EL SISTEMA SIN FALLAR

3. SISTEMA BILINEAL





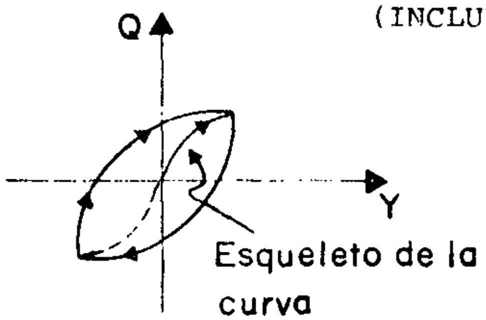
CON ENDURECIMIENTO

SE USA COMO MODELO PARA ANALISIS DE PUENTES COLGANTES

CON ABLANDAMIENTO

SE USA COMO MODELO DE SISTEMAS QUE SE DEGRADAN POR AGRIETAMIENTO (MUROS DE MAMPOSTERIA, POR EJEM)

4. TIPO MASING



(INCLUYE A LOS ANTERIORES COMO CASOS ESPECIALES)

$$\frac{Q - Q_o}{2} = Q_1\left(\frac{y - y_o}{2}\right)$$

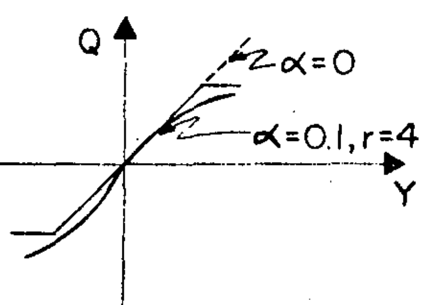
Q_o = FUERZA EN $y = y_o$

y_o = DESPLAZAMIENTO EN EL CUAL EL PROCESO SE INVIRTIO (y CAMBIO DE SIGNO) POR ULTIMA VEZ

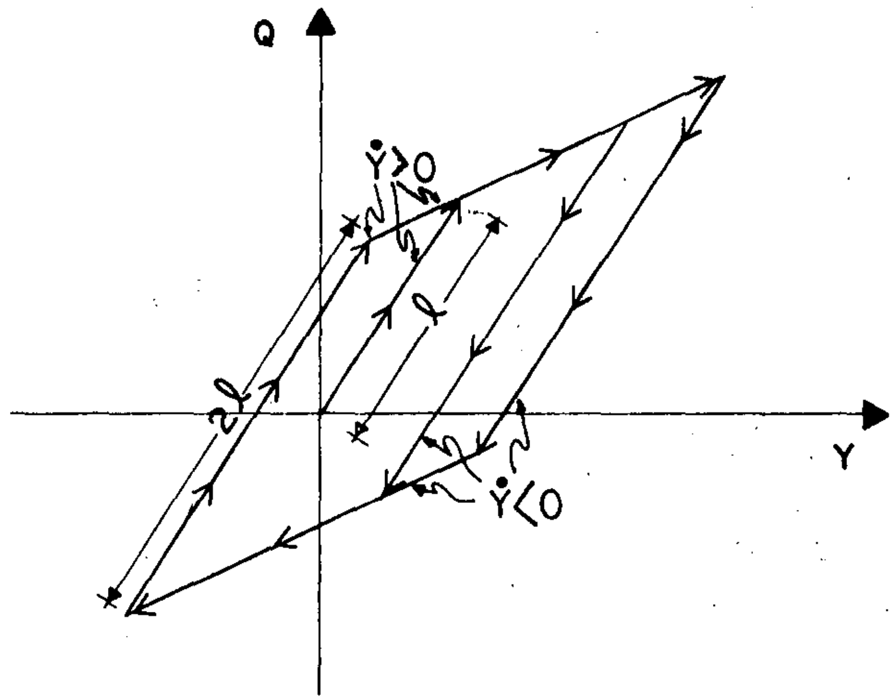
CASO PARTICULAR DEL ESQUELETO

$$\frac{y}{y_1} = \frac{Q}{Q_1} + \alpha\left(\frac{Q}{Q_1}\right)^r \qquad \text{(MODELO RAMBER - OSGOOD)}$$

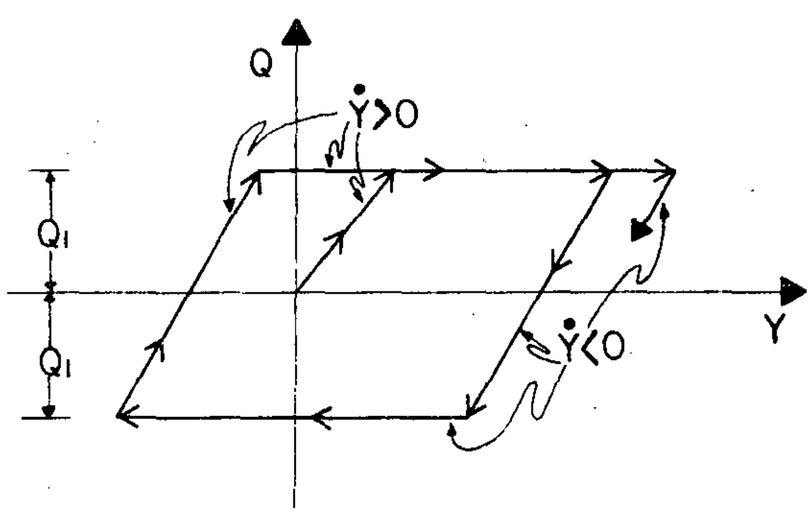
DONDE y_1, Q_1, α y r SON CONSTANTES POSITIVAS



EJEMPLO: CASO BILINEAL



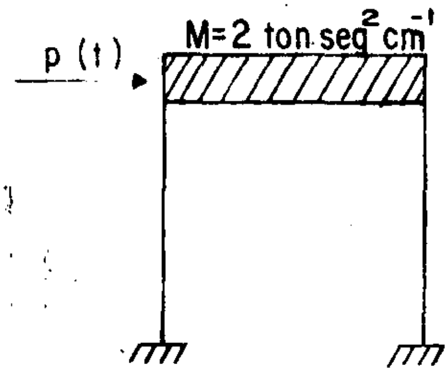
EJEMPLO CASO ELASTOPLASTICO

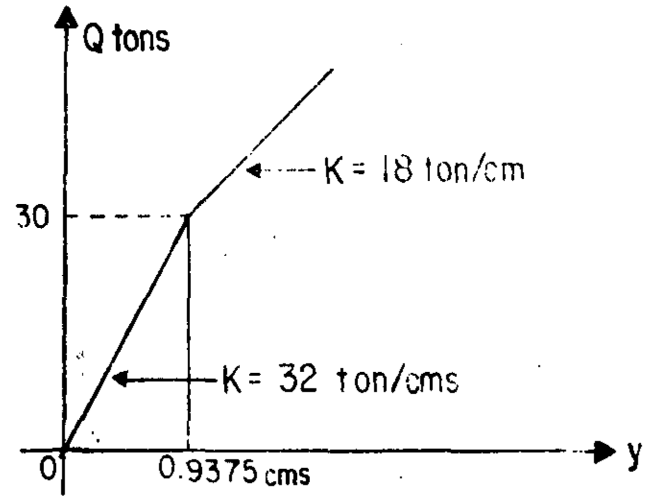


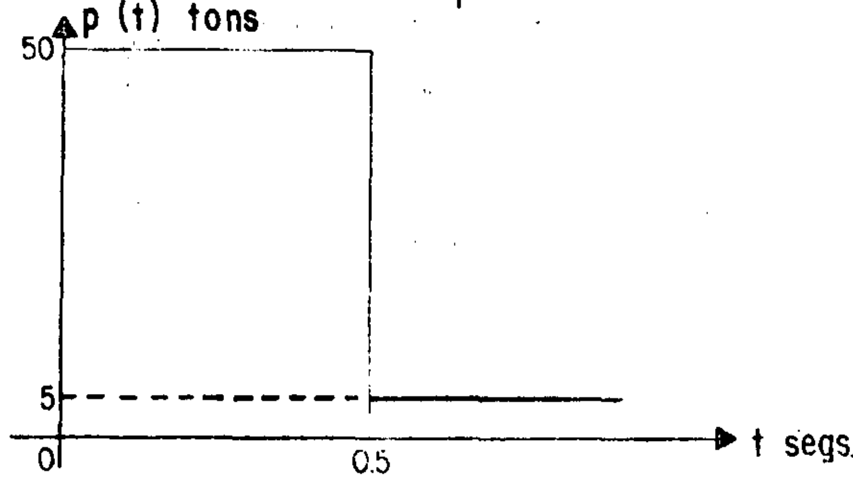
METODO β DE NEWMARK

PARA EL ANALISIS DE SISTEMAS NO LINEALES SE PUEDE USAR EL METODO β DE NEWMARK DESCRITO ANTERIORMENTE.

EJEMPLO







ECUACION DE EQUILIBRIO DINAMICO , $M\ddot{Y} + Q(Y) = P(t)$

$$\ddot{Y} = \frac{P(t) - Q(Y)}{M} = \frac{P(t) - Q(y)}{2} \qquad \text{(I)}$$

PARA LA APLICACION DEL METODO DE NEWMARK SE TIENEN LAS SIGUIENTES EXPRESIONES:

$$t_{i+1} = t_i + At$$

$$\dot{Y}_{i+1} = \dot{Y}_i + (\ddot{Y}_i + \ddot{Y}_{i+1})\ \Delta t/2$$

$$Y_{i+1} = Y_i + \dot{Y}_i\ At + (0.5 - \beta)\ddot{Y}_i\ (\Delta t)^2 + \beta\ \ddot{Y}_{i+1}\ (\Delta t)^2$$

CONSIDERANDO $\Delta t = 0.10$ SEG. Y $\beta = 1/6$ SE PUEDE ESCRIBIR;

$$\dot{Y}_{i+1} = \dot{Y}_i + \frac{1}{20}(\ddot{Y}_i + \ddot{Y}_{i+1}) \qquad (II)$$

$$Y_{i+1} = Y_i + \dot{Y}_i(0.10) + \frac{1}{600}(2\ddot{Y}_i + \ddot{Y}_{i+1}) \qquad (III)$$

EL PROCEDIMIENTO DE CALCULO ES COMO SIGUE:

SE ASUME $\ddot{Y}_{i+1}$

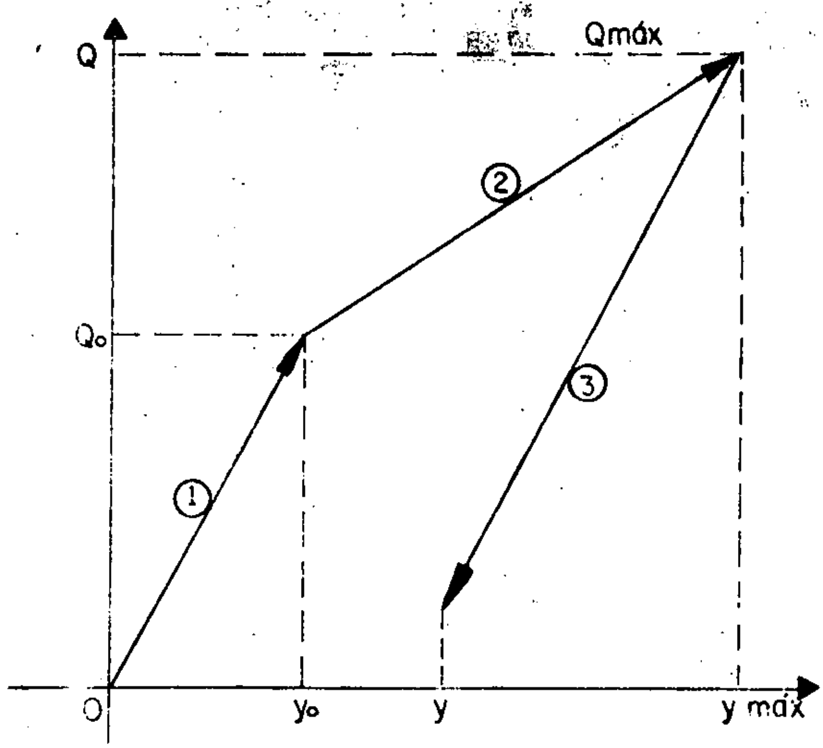
SE CALCULA $\dot{Y}_{i+1}$ CON LA ECUACION (II)

SE CALCULA Y_{i+1} CON LA ECUACION (III)

SE CALCULA UN MEJOR VALOR DE $\ddot{Y}_{i+1}$ CON LA ECUACION (I).

ETC.

PARA LA FUNCION DE RESISTENCIA Q SE TIENEN LOS SIGUIENTES CASOS:



1. COMPORTAMIENTO ELASTICO , $Q = 32\ Y$ TONS

2. CAMBIO DE RIGIDEZ , $Q = 30 + 18\ (Y-Y_o)$ TON

3. DESCARGA , $Q = Q_{máx} - 32(Y_{MAX}-Y)$ TONS

ESTA ULTIMA EXPRESION MANTIENE SU VALIDEZ HASTA QUE, $(Y_{MAX}-Y) \leq 2Y_o$

$Y_o = 0.9375$ CMS ; $Q_o = 30.0$ TON

PARA t = 0, $\ddot{y} = \frac{P}{M} = \frac{50}{2} = 25$; $y = 0$; $\dot{y} = 0$

PARA t = 0.10, $y_i = \dot{y}_i = 0$; $\ddot{y}_i = 25$

1er. CICLO

SEA $\ddot{y}_{i+1} = 20$ COMO PRIMER TANTEO. EN TAL CASO

$$\dot{y}_{i+1} = 0 + \frac{1}{20} (20 + 25) = 2.25$$

$$y_{i+1} = \dot{0} + 0.10 \times 0 + \frac{1}{600} (2 \times 25 + 20) = 0.1167$$

$$Q = 32 \times 0.1167 = 3.7330$$

$$\ddot{y}_{i+1} = \frac{50 - 3.733}{2} = 23.134$$

2o. CICLO

$$\dot{y}_{i+1} = 23.134/2 = 16.567$$

$$y_{i+1} = 73.134/600 = 0.1219$$

$$Q = 32 \times 0.1219 = 3.9000$$

$$\ddot{y}_{i+1} = (50 - 3.9)/2 = 23.050$$

3er. CICLO

.

.

.

4o. CICLO

$$\ddot{y}_{i+1} = 23.052$$

$$\dot{y}_{i+1} = 23.052/2 = 2.4026$$

$$y_{i+1} = 73.052/600 = 0.12175$$

$$Q = 32 \times 0.12175 = 3.8960$$

$$\ddot{y} = (50 - 3.8960)/2 = 23.052 \quad \ldots \text{ ETC.}$$

LOS CALCULOS BASICOS SE MUESTRAN EN LA TABLA SIGUIENTE:

t SEGS	P TONS	$\ddot{Y}$ CM SEG^{-2}	$\dot{Y}$ CM SEG^{-1}	Y CMS	Q TONS
0.0	50.00	25.000	0.00	0.00	0.00
0.10	50.00	20.000	2.2500	0.1167	3.7330
		23.134	2.4070	0.1219	3.9000
		23.050	2.4025	0.12175	3.3960
		23.052	2.4026	0.12175	3.8960
0.20	50.00	20.000	4.5552	0.4722	15.110
		17.445	4.4270	0.46793	14.970
		17.513	4.4310	0.46804	14.977
		17.511	4.43075	0.46204	14.977
0.30	50.00	10.000	5.8060	0.98610	30.8750
		9.560	5.7840	0.98540	30.8620
		9.569	5.7848	0.98543	30.8630
0.40	50.00	0.00	6.2630	1.5958	41.349
		4.0750	6.4670	1.6026	41.972
		4.0141	6.4640	1.6025	41.970
		4.0150	6.4640	1.60250	41.970
0.50^{-}	50.00	0.00	6.6650	2.2623	53.846
		-1.9230			
		-1.9000	6.56975	2.2591	53.789
		-1.8944			
		-1.8946	6.5700	2.25912	53.789
0.50+	5.00	-24.3946	6.5700	2.25912	53.[illegible]
0.60	5.00	-30.000	3.8503	2.7848	63.251
		-29.126	3.8940	2.78626	63.278
		-29.136	3.89347	2.78624	63.277
		-29.138	3.89347	2.78624	63.277
0.70	5.00	-32.000	0.83657	3.025127	67.577
		-31.289			
		-31.320	0.87057	3.02626	67.5[illegible]
		-31.299			
		-31.301	0.87147	3.02641	67.600
0.7278	5.00	-31.620	-0.00313	3.03850	67.810
		-31.409			
		-31.420	-0.000352	3.03853	67.8[illegible]
		-31.4093	-0.000205	3.03853	67.8[illegible]

En t=0.5 + SEG, $\Delta\ddot{y}$ =-45/2 = -22.5 ∴ -22.5 - 1.9946 = -24.3946

CONTINUACION DEL CUADRO ANTERIOR

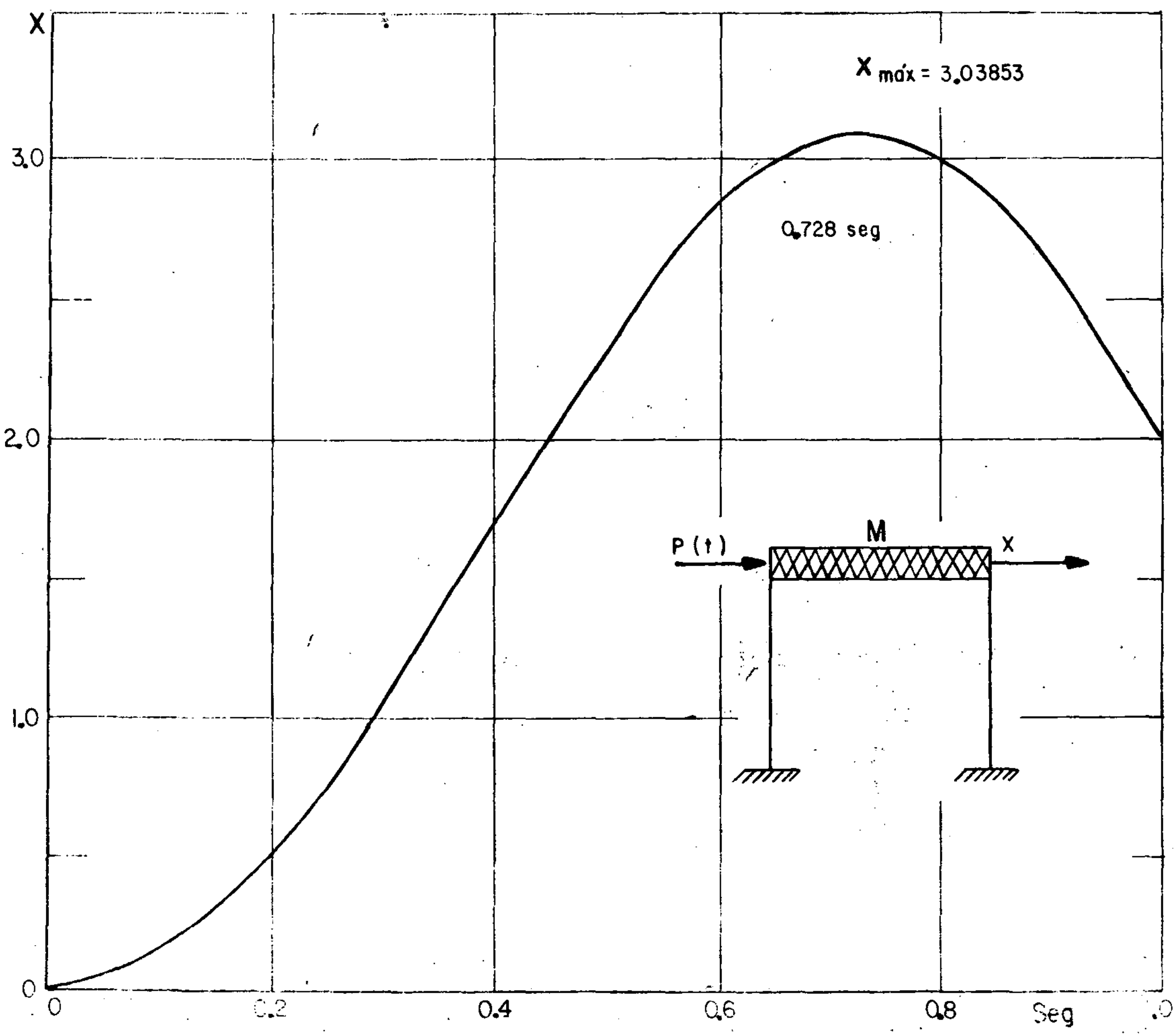
t	p	$\ddot{Y}$	$\dot{Y}$	Y	Q
0.80	5.0	-28.000	-2.1449	2.959611	65.293
		-30.146			
		-30.000	-2.21708	2.957874	65.237
		-30.118			
		-30.117	-2.22127	2.95777	65.234
0.90	5.0	-27.00	-5.07712	2.59025	53.473
		-24.236			
		-25.00	-4.97712	2.59358	53.580
		-24.290			
		-24.294	-4.94182	2.59476	53.617
		-24.308	-4.94242	2.59474	53.617
1.00	5.0	-14.00	-6.85782	1.99614	34.461
		-14.7305			
		-14.7200	-6.89382	1.99494	34.423
		-14.7120	-6.89342	1.99495	34.423

EN ESTOS CALCULOS SE INTRODUJO $t = 0.50^-$ Y 0.50^+ PORQUE PARA ESTE INSTANTE SE PRODUCE UN CAMBIO BRUSCO EN LA CARGA P(t) DE 50.00 TONS A 5.00 TONS, CON LO CUAL SE PRODUCE UN CAMBIO BRUSCO EN LA ACELERACION DEL SISTEMA $\ddot{Y}$. EN ESTE INSTANTE NO SE PRODUCEN CAMBIOS EN $\dot{Y}$ Y Y. EL TIEMPO $t = 0.7273$ SEG. SE INTRODUJO POR LA NECESIDAD DE CALCULAR LOS VALORES DE Y Y DE Q, PUES A PARTIR DE DICHO INSTANTE SE INICIA LA DESCARGA DEL SISTEMA. ESTA CONDICION SE ENCONTRO SOBRE LA BASE DE APROXIMAR $\dot{Y}$ A CERO, OBTENIENDOSE $Y_{MAX}=3.03853$ CMS Y $Q_{MAX} = 67.818$ TON.

EN EL CUADRO SIGUIENTE SE PRESENTA UN RESUMEN DE LOS RESULTADOS.

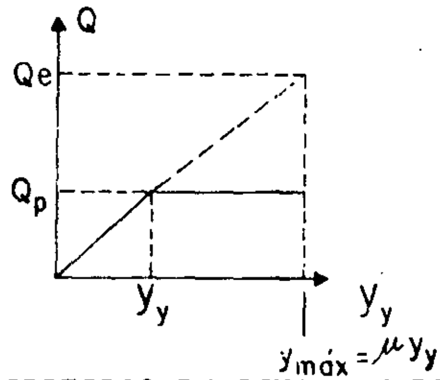
t Seg.	$\ddot{Y}$ (supuesta) Cm Seg^{-2}	P Ton	Y Cm.	Q Ton	$\ddot{Y}$ (calculado) Cm Seg^{-2}	$\dot{Y}$ Cm Seg^{-1}	N O T A S
0.0	- -	50.00	0.00	0.00	25.00	0.00	
0.10	23.0520	50.00	0.12175	3.896	23.0520	2.40260	
0.20	17.5110	50.00	0.46804	14.977	17.5110	4.43075	
0.30	9.5690	50.00	0.98543	30.863	9.5690	5.78480	CAMBIO DE RIGIDEZ
0.40	4.0150	50.00	1.60250	41.970	4.0150	6.4640	
0.50^-	-1.8946	50.00	2.25912	53.789	-1.8946	6.5700	
0.50+	- -	5.00	2.25912	53.789	-24.3945	6.5700	CAMBIO DE CARGA
0.60	-29.1380	5.00	2.78624	63.277	-29.1380	3.89347	
0.70	-31.3010	5.00	3.02641	67.600	-31.3010	0.87147	
0.7278	-31.4093	5.00	3.03853	67.818	-31.4093	-0.000205	Qmáx, Ymáx.
0.800	-30.1170	5.00	2.95777	65.234	-30.1170	-2.22127	
0.90	-24.3080	5.00	2.59474	53.617	-24.3080	-4.94242	
1.00	-14.7120	5.00	1.99495	34.423	-14.7120	-6.89342	

RESPUESTA MAXIMA $\begin{cases} V \text{ máx} = 3.03853 \text{ cms} \\ Q \text{ máx} = 67.818 \text{ tons} \end{cases}$


X
X máx = 3.03853
0.728 seg
3.0
2.0
1.0
0
0.2
0.4
0.6
0.8
Seg
M
P (t)
X

CRITERIOS PARA TRAZAR ESPECTROS DE DISEÑO ELASTOPLASTICOS A PARTIR DEL ELASTICO

1. CRITERIO DE IGUAL DESPLAZAMIENTO MAXIMO DEL SISTEMA ELASTICO Y EL ELASTOPLASTICO DE IGUAL PERIODO:

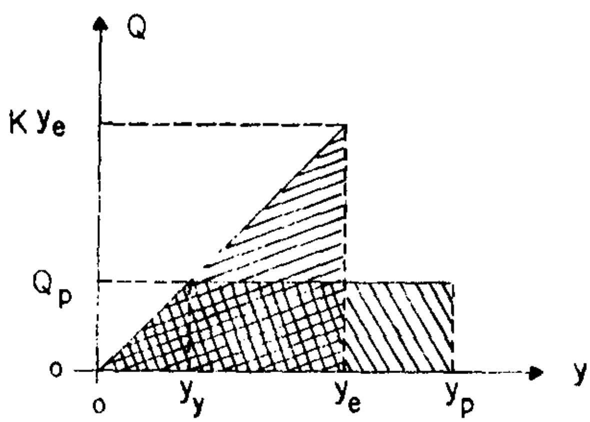


$$Q_p = Ky_y = \frac{Ky_{MAX}}{\mu} = \frac{Q_e}{\mu}$$

$$y_{MAX} = D_e = \mu y_y = \mu D_p$$

$$D_p = \frac{De}{\mu}$$

2. CRITERIO DE IGUAL ENERGIA ABSORVIDA POR LA ESTRUCTURA:



$$\frac{Ky_e y_e}{2} = \frac{Ky_y y_y}{2} + Ky_y(y_p - y_y)$$

$$\frac{1}{2}y_e^2 = \frac{1}{2}y_y^2 + y_y y_p - y_y^2 = y_y y_p - \frac{y_y^2}{2}$$

$$\frac{1}{2}\left(\frac{y_e}{y_y}\right)^2 = \frac{y_p}{y_y} - \frac{1}{2} = \mu - \frac{1}{2}$$

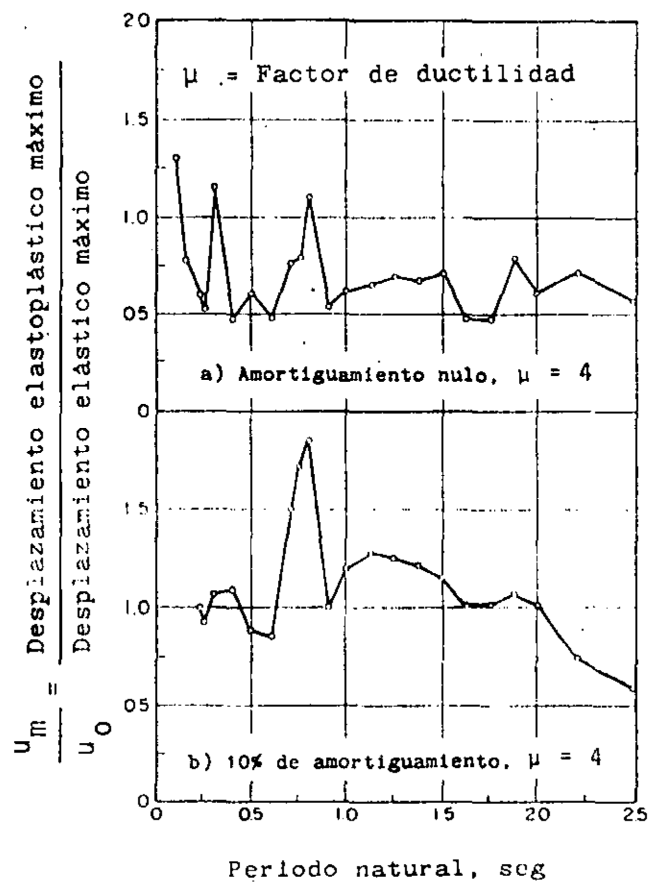
$$\frac{y_e}{y_y} = \sqrt{2\mu - 1}$$

$$y_y = \frac{y_e}{\sqrt{2\mu - 1}}$$

$$y_{y\ MAX} = D_p = \frac{y_e|_{MAX}}{\sqrt{2\mu - 1}} = \frac{D_e}{\sqrt{2\mu - 1}}$$

POR LO TANTO

$$D_p = D_e/\sqrt{2\mu - 1} \qquad Y \qquad Q_p = Q_e/\sqrt{2\mu - 1}$$



Comparación de la respuesta máxima de un sistema elastoplástico y uno elástico. Sismo de El Centro, Cal. (1940). Según Blume, Newmark y Corning.

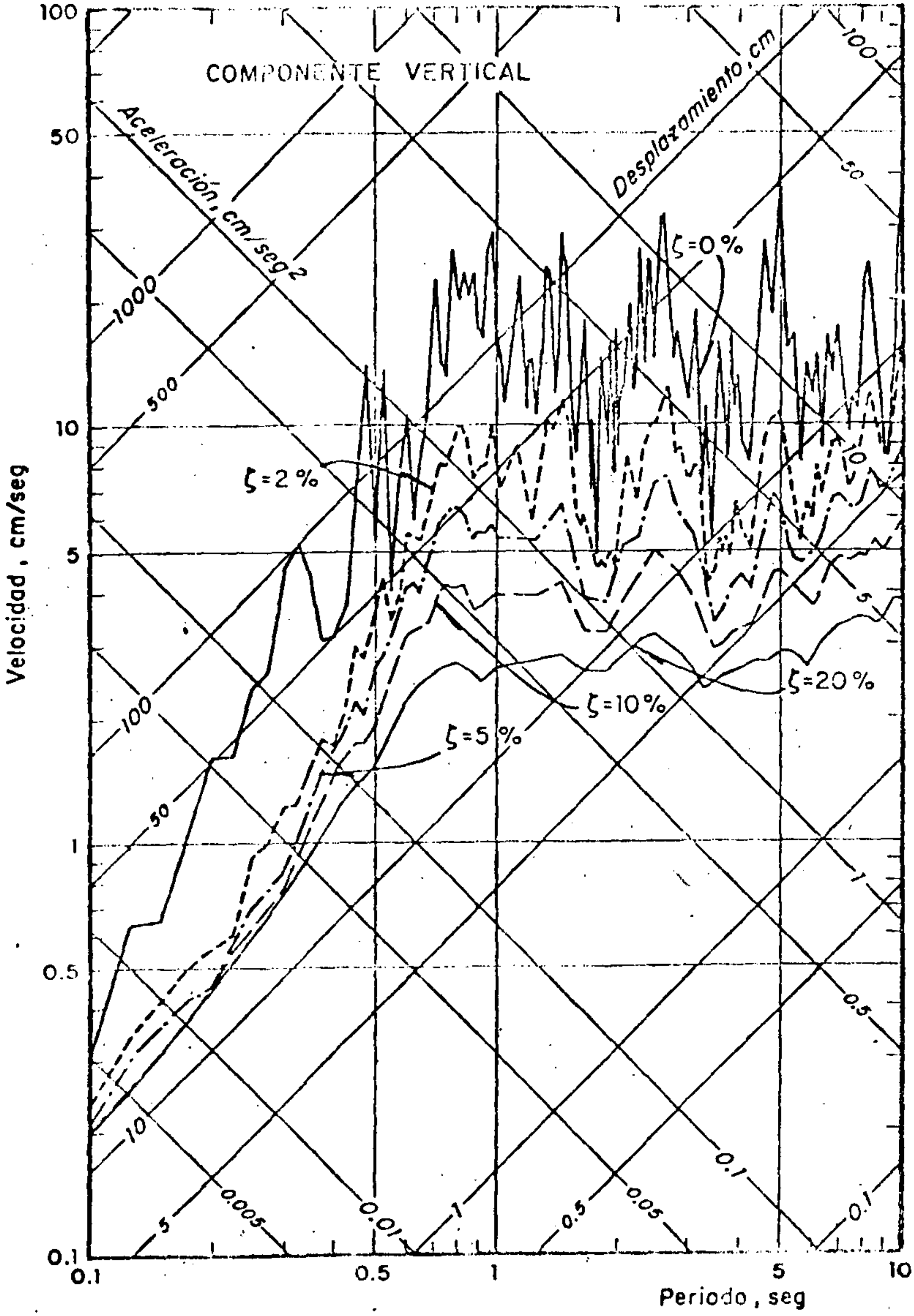


Fig 82 Espectros de respuesta. Alameda Central,
11 de mayo de 1962

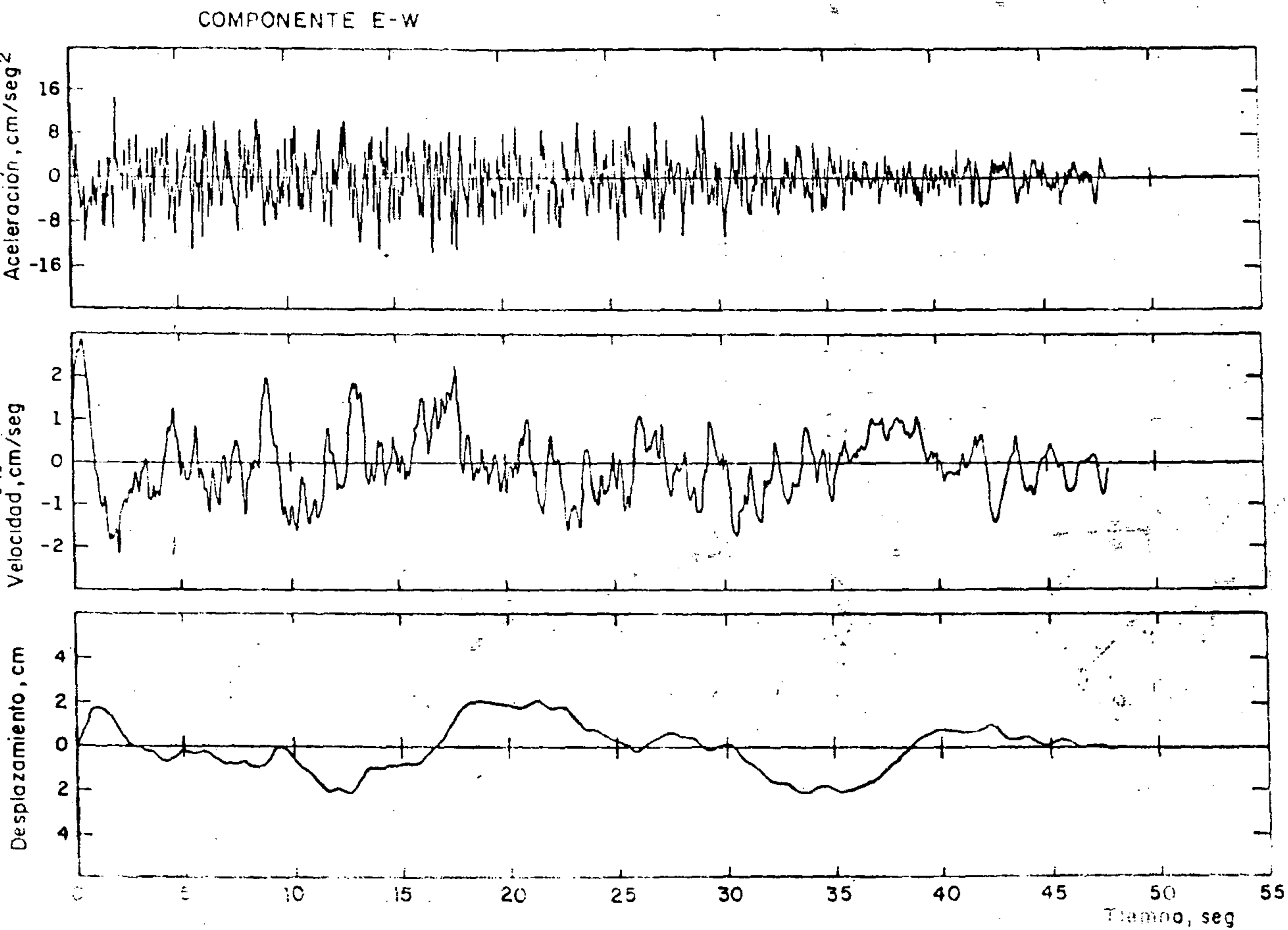


Fig. 2 Movimiento del terreno. Ciudad Universit[illegible], 6 de julio de 1964

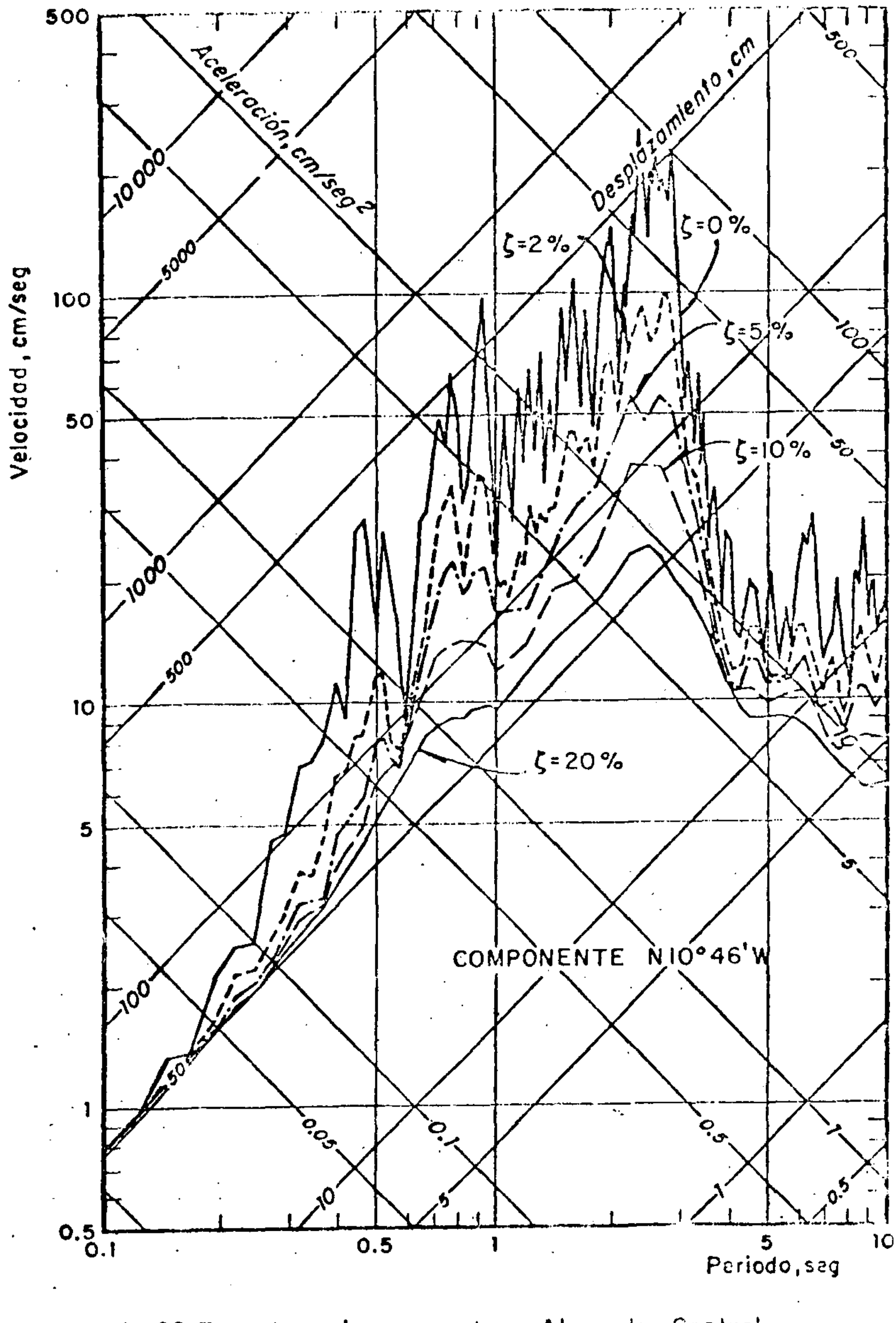


Fig 80 Espectros de respuesta. Alameda Central,
11 de mayo de 1962

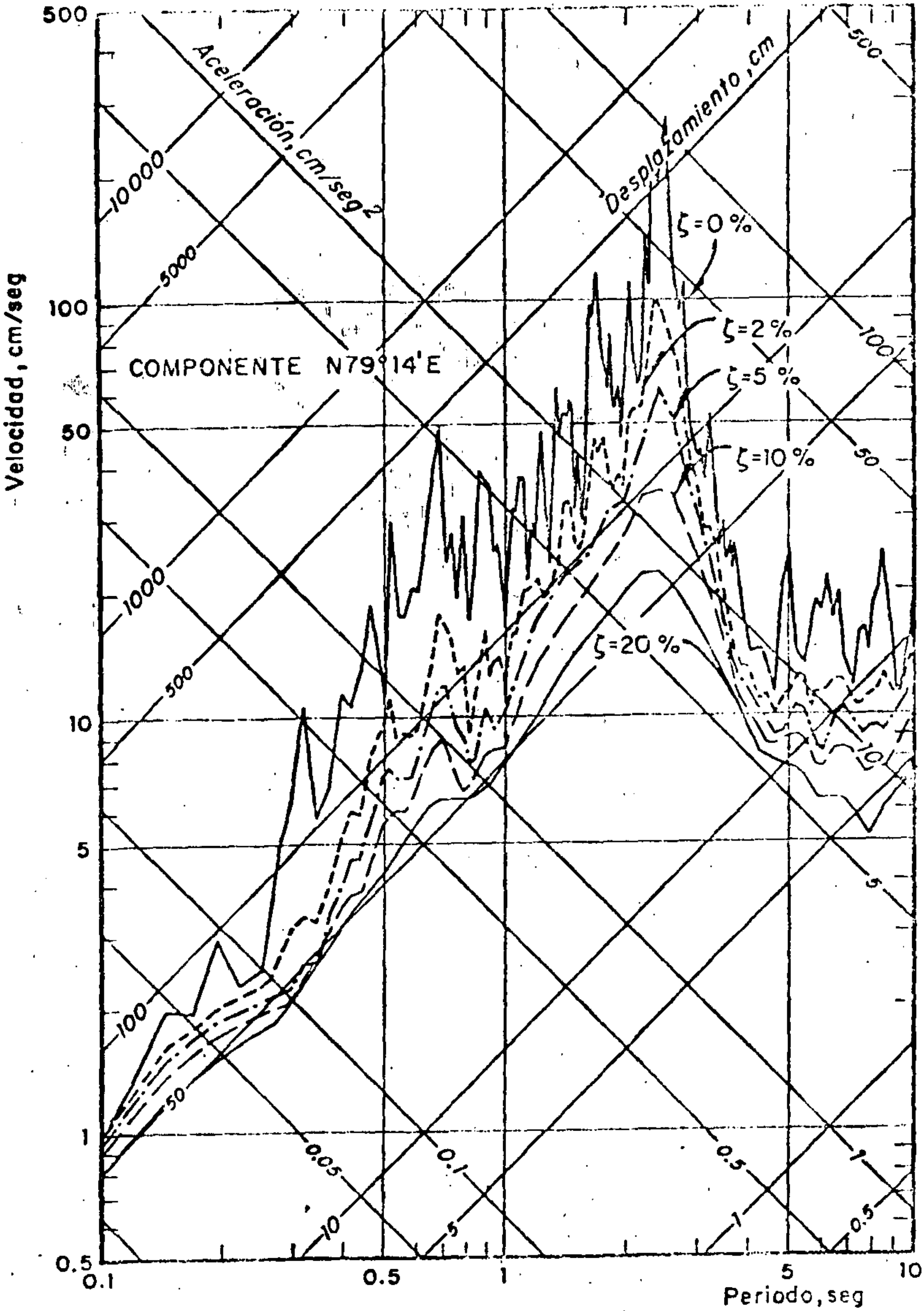


Fig 81 Espectros de respuesta. Alameda Central, 11 de mayo de 1962

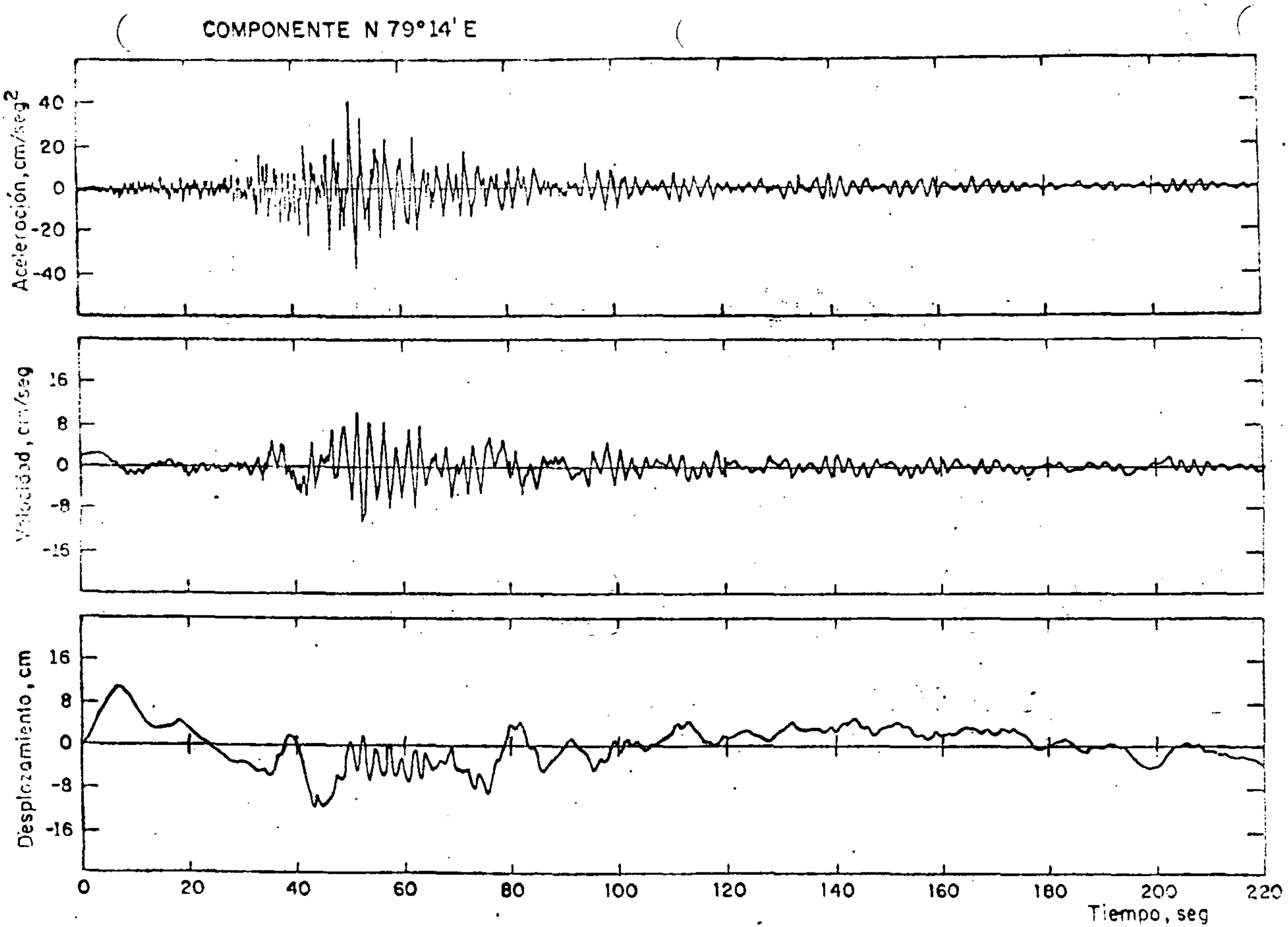


Fig 8 Movimiento del terreno. Alameda Central, 11 de mayo de 1962

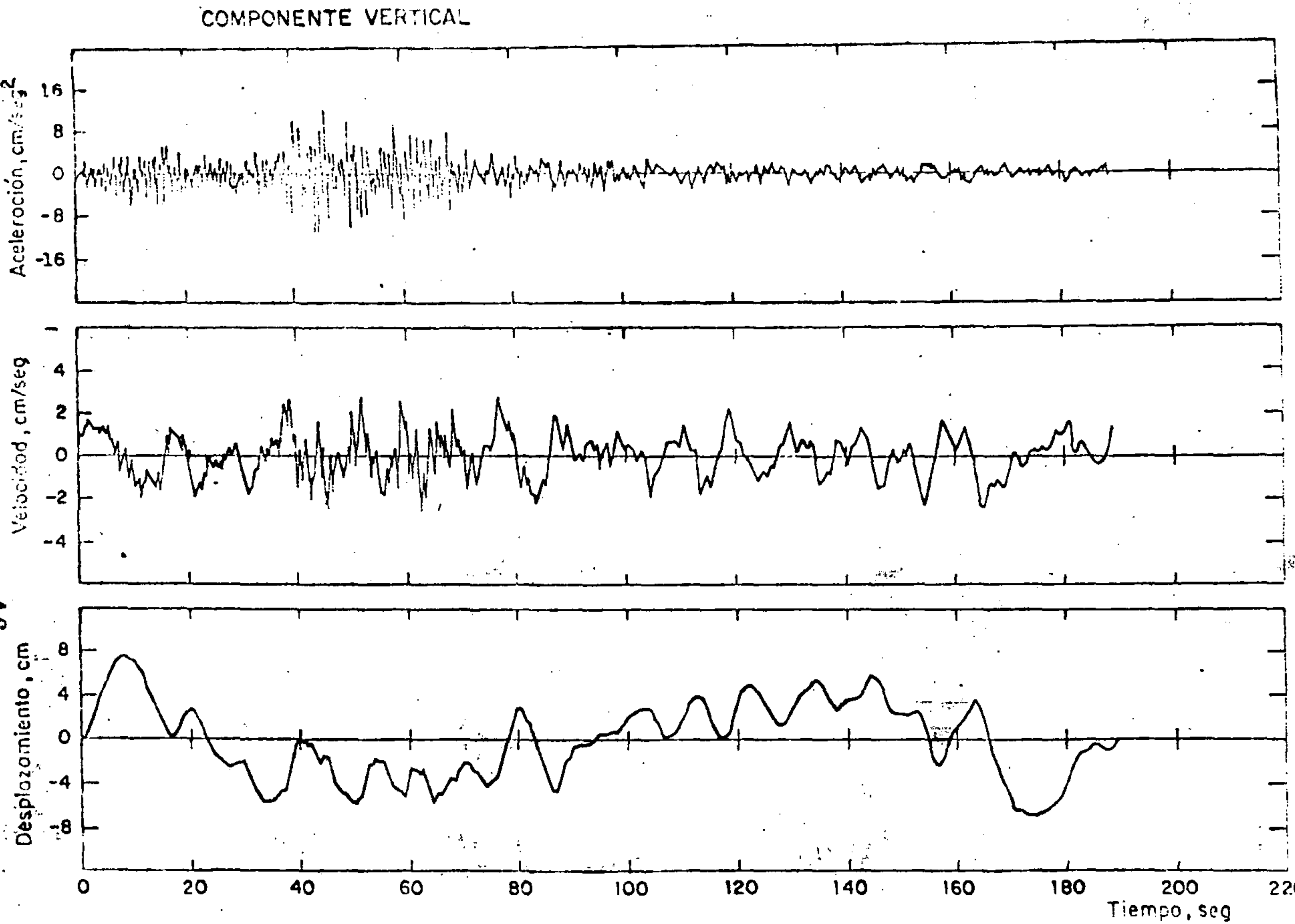
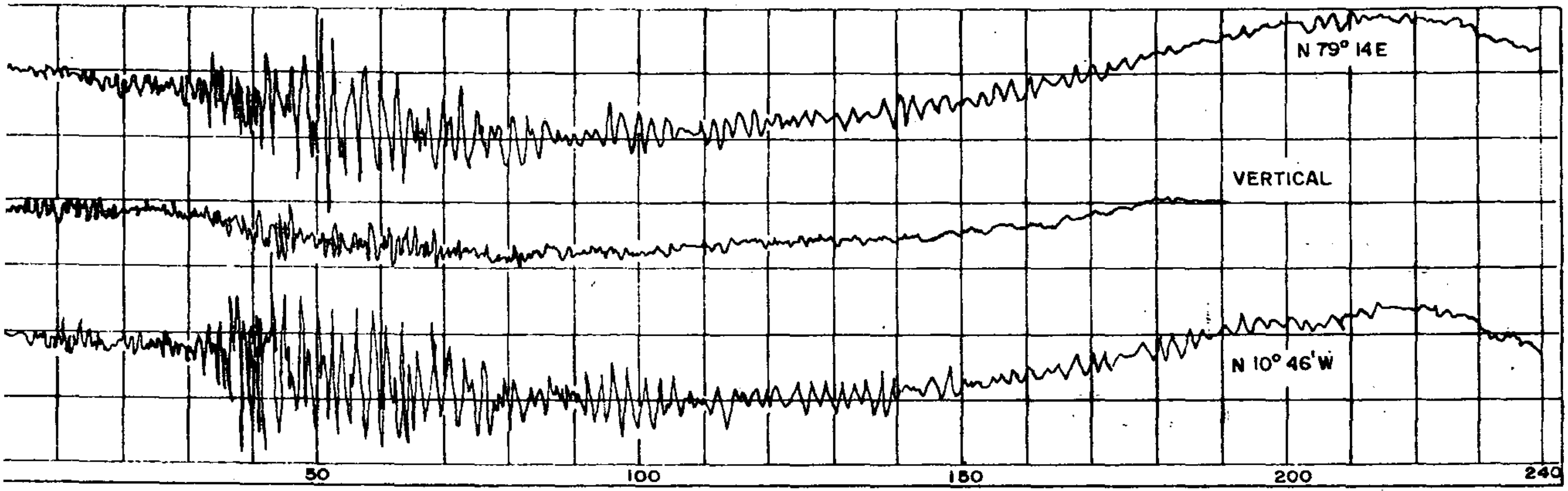
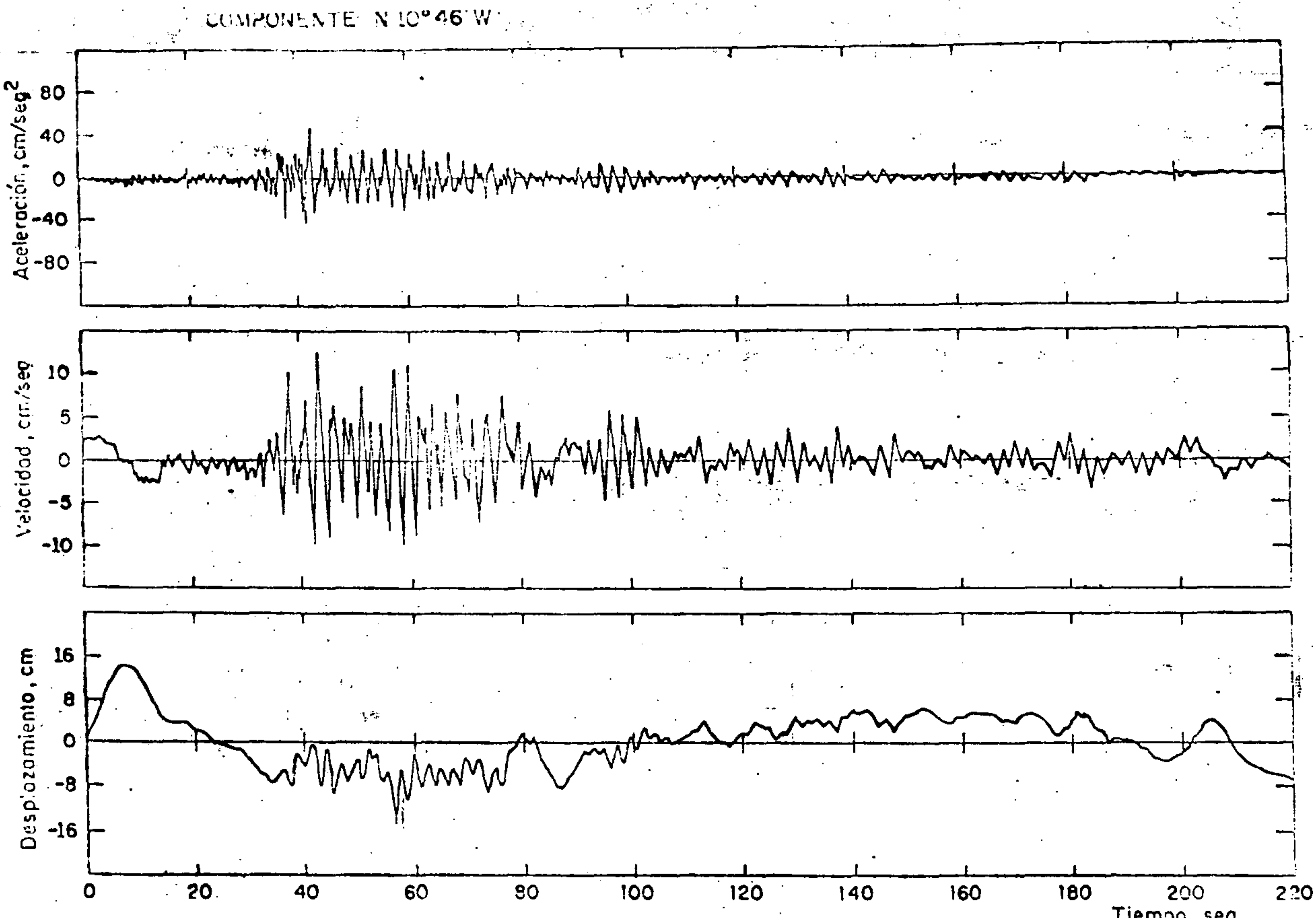


Fig 9 Movimiento del terreno. Alameda Central, 11 de mayo de 1962

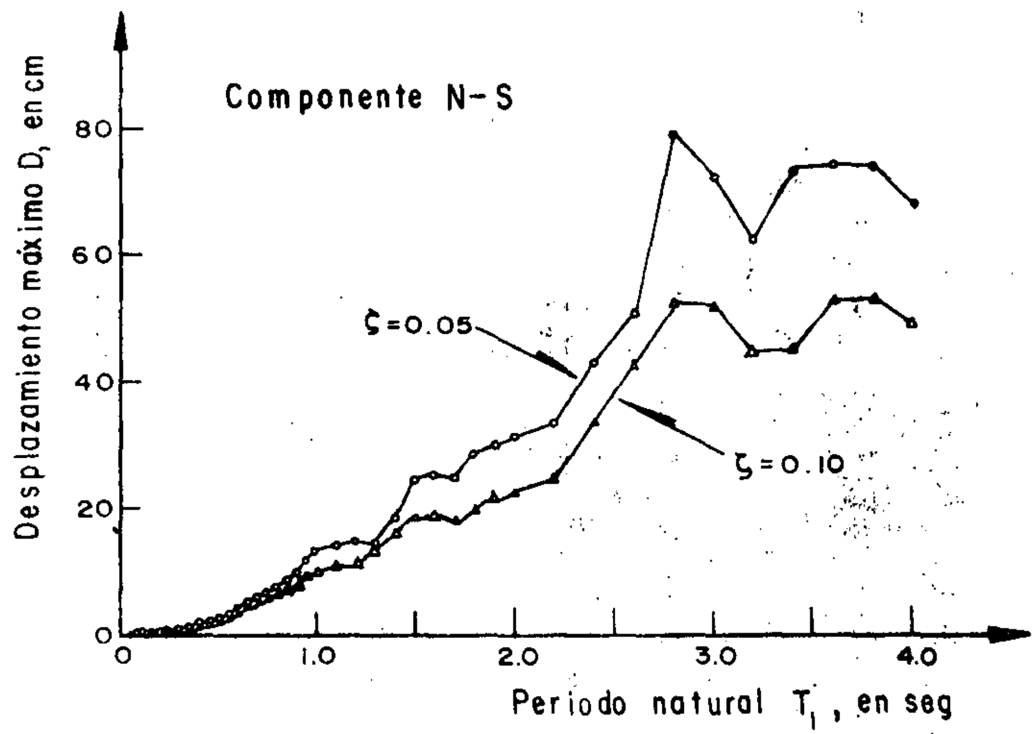


Acelerogramas originales del sismo registrado el
11-V-1962, en la ALAMEDA CENTRAL, Mex. D.F.
(Tomada de la ref 2)

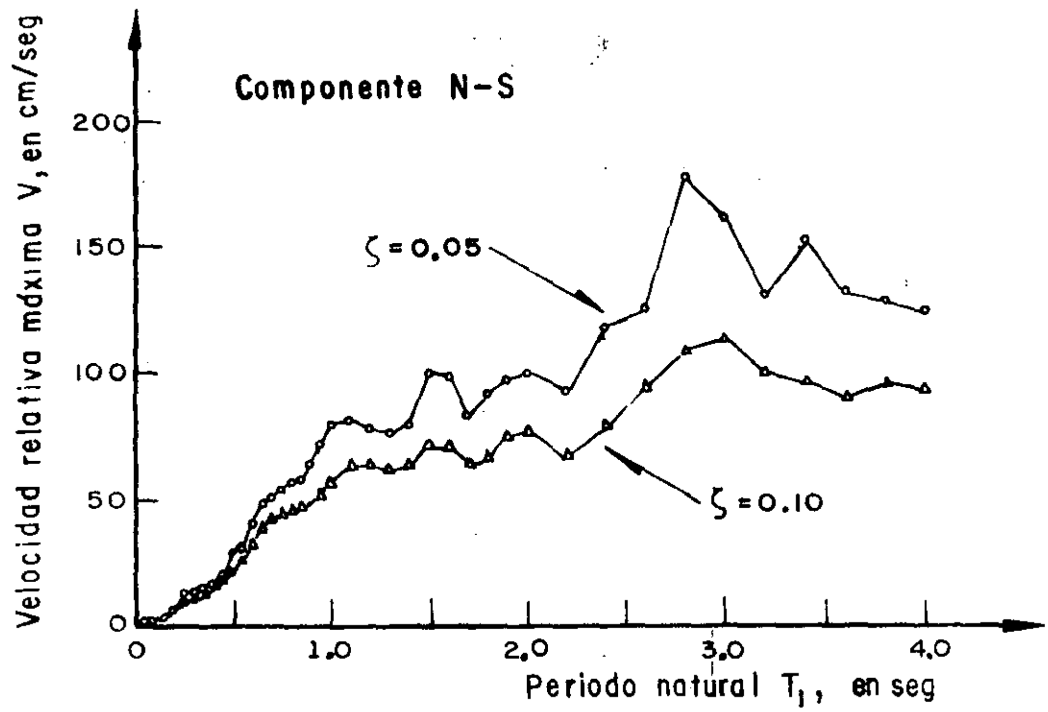


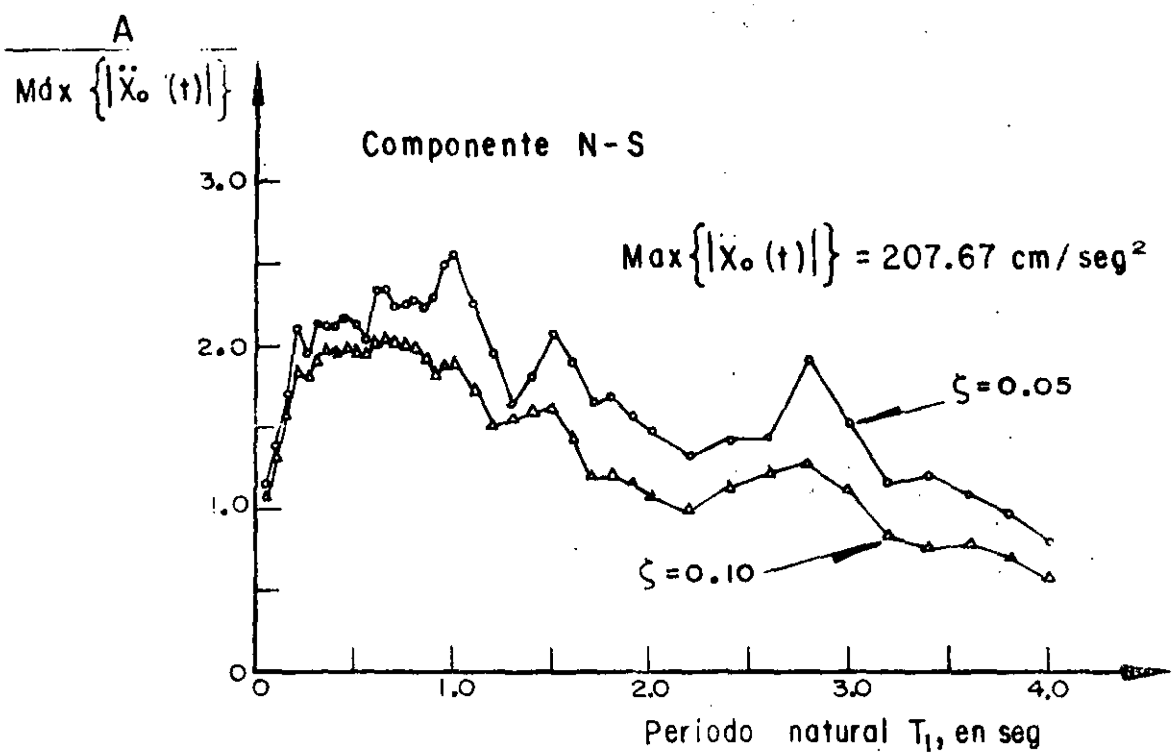
44

Fig 7   Movimiento del terreno. Alameda Central, 11 de mayo de 1962



Espectro de desplazamientos. Sismo de Tokachi-Oki, Japón (1968). Según H. Tsuchida, E. Kurata y K. Sudo, ref 4



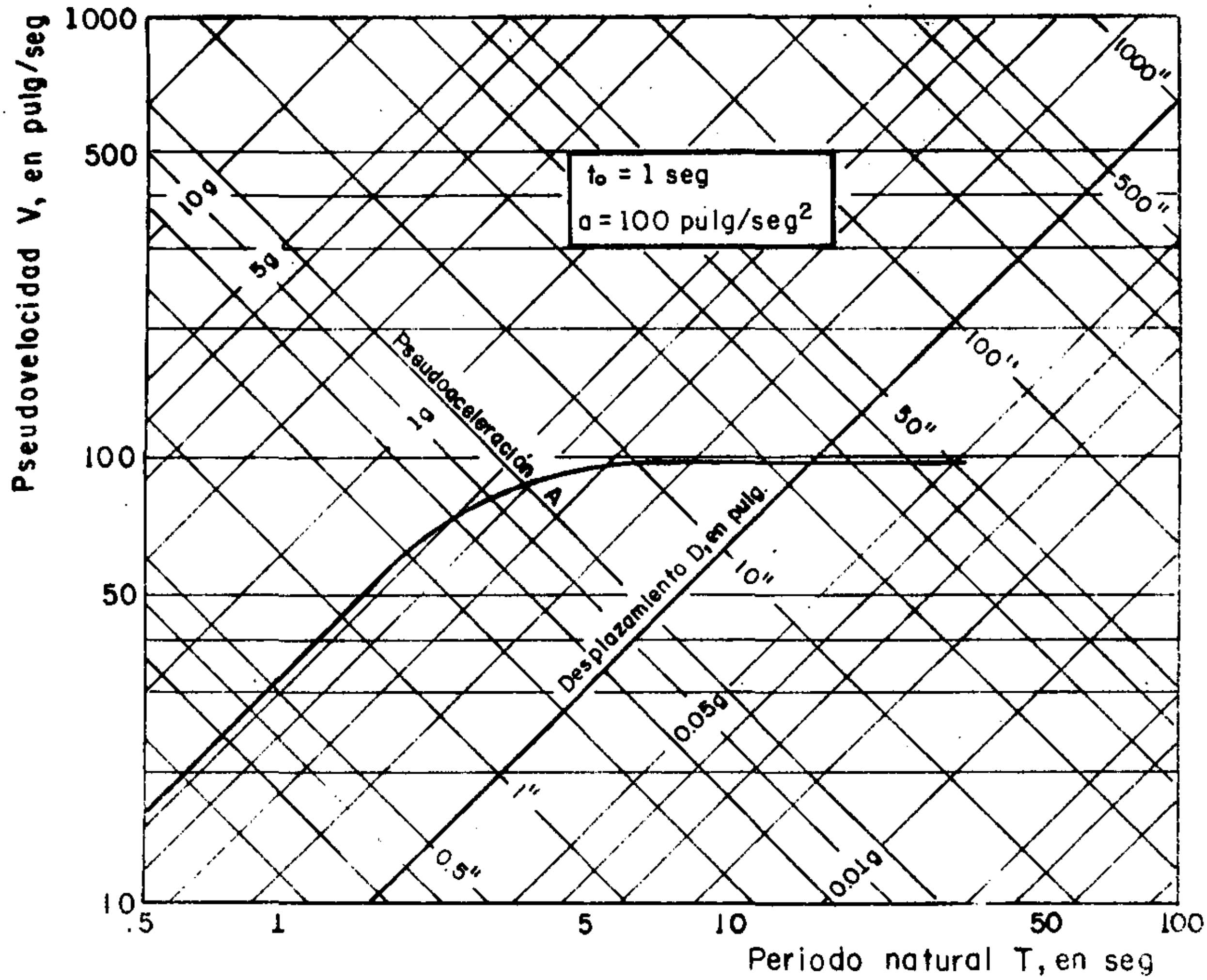


Espectros de velocidades y de aceleraciones. Sismo de Tokachi-Oki, Japón (1968). Según H. Tsuchida, E. Kurata y K. Sudo, ref. 4

$$S_V \quad \frac{100T}{\pi}\left|\operatorname{sen}\frac{\frac{2\pi}{T}\times 1}{2}\right|$$

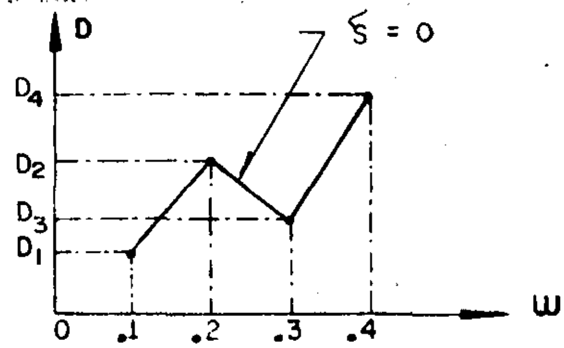
$$= \frac{100T}{\pi}\left|\operatorname{sen}\frac{\pi}{T}\right| \quad \text{SI } T>2 \text{ SEG}$$

$$\lim_{T\to\infty} S_V = 100 \text{ IN/SEG}$$



Espectro no amortiguado correspondiente a un pulso rectangular de aceleraciones. Según N. Newmark y E. Rosenblueth, ref 1

EN TAL CASO, LA GRAFICA



ES EL *ESPECTRO DE RESPUESTA DE DESPLAZAMIENTOS* PARA $\zeta=0$. SI ESTE PROCESO DE REPITE FIJANDO OTROS VALORES DE ζ, POR EJEMPLO, $\zeta=0.02$, 0.05, 0.1, 0.2, ETC, SE OBTENDRAN LOS ESPECTROS DE DESPLAZAMIENTOS CORRESPONDIENTES.

DE MANERA ANALOGA SE PUEDEN OBTENER LOS ESPECTROS PARA OTROS TIPOS DE RESPUESTA, TALES COMO *VELOCIDAD RELATIVA*, *ACELERACION ABSOLUTA*, ETC, QUE SON, RESPECTIVAMENTE

$$V = \text{MAX}\left|\dot{y}(t)\right|_{\zeta,\omega} \quad ; \quad A = \text{MAX}\left|\ddot{x}(t)\right|_{\zeta,\omega} \tag{29}$$

PSEUDO - ESPECTROS

ESTADISTICAMENTE SE HA ENCONTRADO QUE

$$S_V = \omega D \doteq V \tag{30}$$

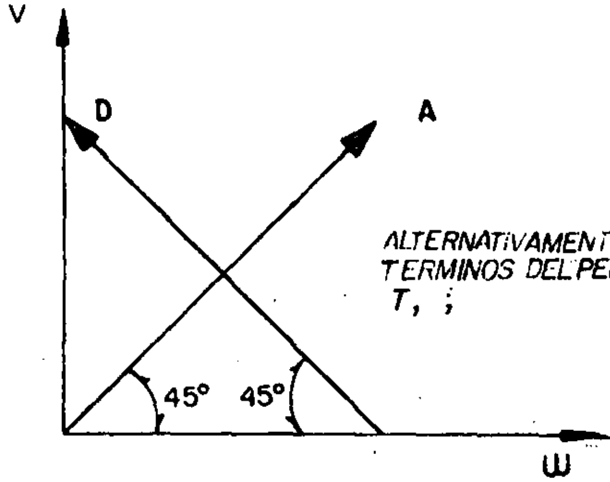
$$S_A = \omega^2 D \doteq A \doteq \omega V \tag{31}$$

A S_V Y S_A SE LES LLAMA *PSEUDOESPECTROS*.

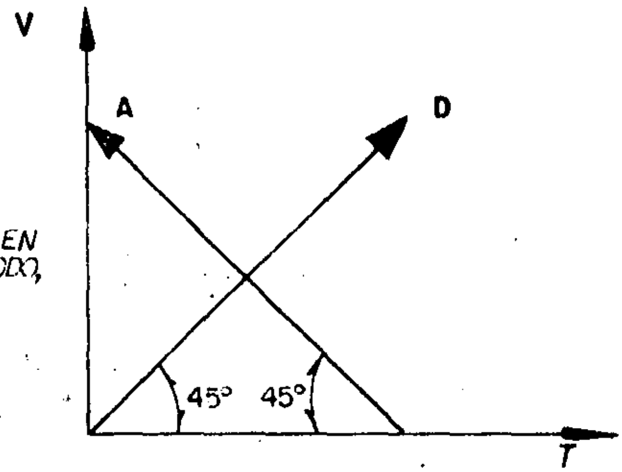
DE LA EC. (30): $\log D = \log V - \log \omega = \log V + \log T - \log 2\pi$

DE LA EC. (31): $\log A = \log V + \log \omega = \log V - \log T + \log 2\pi$

ESTAS ECUACIONES CORRESPONDEN A LINEAS RECTAS EN PAPEL LOGARITMICO; LA PRIMERA CON PENDIENTE -1 Y LA SEGUNDA CON PENDIENTE +1, SI SE USA ω COMO VARIABLE INDEPENDIENTE; SI SE USA T, LA PRIMERA TENDRA PENDIENTE + 1, Y LA SEGUNDA, -1.

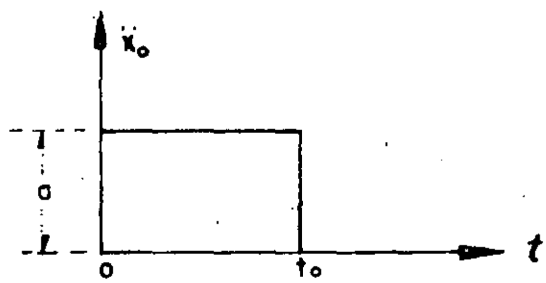


ALTERNATIVAMENTE, EN TERMINOS DEL PERIODO, T, ;



EJEMPLO

CALCULAR EL ESPECTRO CORRESPONDIENTE A LA EXCITACION (CONSIDERESE $\xi=0$)



EN UN EJEMPLO ANTERIOR SE OBTUVO

$$y(t) = \frac{-a}{\omega^2}(1 - \cos\omega t), \text{ SI } 0 \le t \le t_o$$

$$D = \text{MAX}|Y(t)| = \frac{2a}{\omega^2} \; ; \; 0 \le \frac{T}{2} \le t_o, \; (0 \le T \le 2t_o)$$

$$S_V = \omega D = \frac{2a}{\omega} \; , \; S_A = \omega V = 2a$$

Y $$D = \text{MAX}|y(t)| = \frac{2a}{\omega^2} \text{ sen } \frac{\omega t_o}{2} \; , \; \text{SI } T > 2t_o$$

$$S_V = \omega D = \frac{2a}{\omega} \left|\text{sen } \frac{\omega t_o}{2}\right| \; ; \; S_A = \omega V = 2a\left|\text{sen} \frac{\omega t_o}{2}\right|$$

$$\lim_{\omega \to 0} S_V = \lim_{\omega \to 0} \left\{ a t_o \frac{\text{sen} \frac{\omega t_o}{2}}{\frac{\omega t_o}{2}} \right\} = a t_o$$

CASO PARTICULAR: SI $t_o = 1$ SEG y $a = 100$ IN/SEG^2

$$S_V = \frac{2 \times 100}{\frac{2\pi}{T}} = \frac{100}{\pi} T \; , \; \text{SI } 0 \le T \le 2 \text{ SEG}$$

ESPECTROS DE RESPUESTA ESTRUCTURAL.

RECORDEMOS QUE LA SOLUCION DEL PROBLEMA DE VIBRACIONES FORZADAS CON EXCITACION SISMICA ES

$$Y(t) = \frac{-1}{\omega'} \int_{-\infty}^{t} \ddot{x}_o(t-\tau) e^{-\zeta\omega(t-\tau)} \operatorname{sen} \omega'(t-\tau) d\tau$$

DE LA OBSERVACION DE ESTA ECUACION SE CONCLUYE QUE EL DESPLAZAMIENTO RELATIVO, $Y(t)$, ES FUNCION DEL TIEMPO, t. EL AMORTIGUAMIENTO, ζ, Y LA FRECUENCIA CIRCULAR NATURAL, ω (O DEL PERIODO NATURAL):

$y(t) = f(t, \omega, \zeta)$.

FIJEMOS UN VALOR DE ζ, POR EJEMPLO $\zeta=0$, Y LUEGO ASIGNEMOS VALORES A ω, POR EJEMPLO 0.1, 0.2. 0.3, ETC, HASTA CUBRIR UN INTERVALO DE INTERES, Y PARA CADA CASO CALCULEMOS LA FUNCION RESULTANTE DE APLICAR LA ECUACION ANTERIOR. CON ESTA OBTENEMOS

$$y_1(t) = f_1(t,\ 0.1,\ 0) = f_1(t)$$

$$y_2(t) = f_2(t,\ 0.2,\ 0) = f_2(t)$$

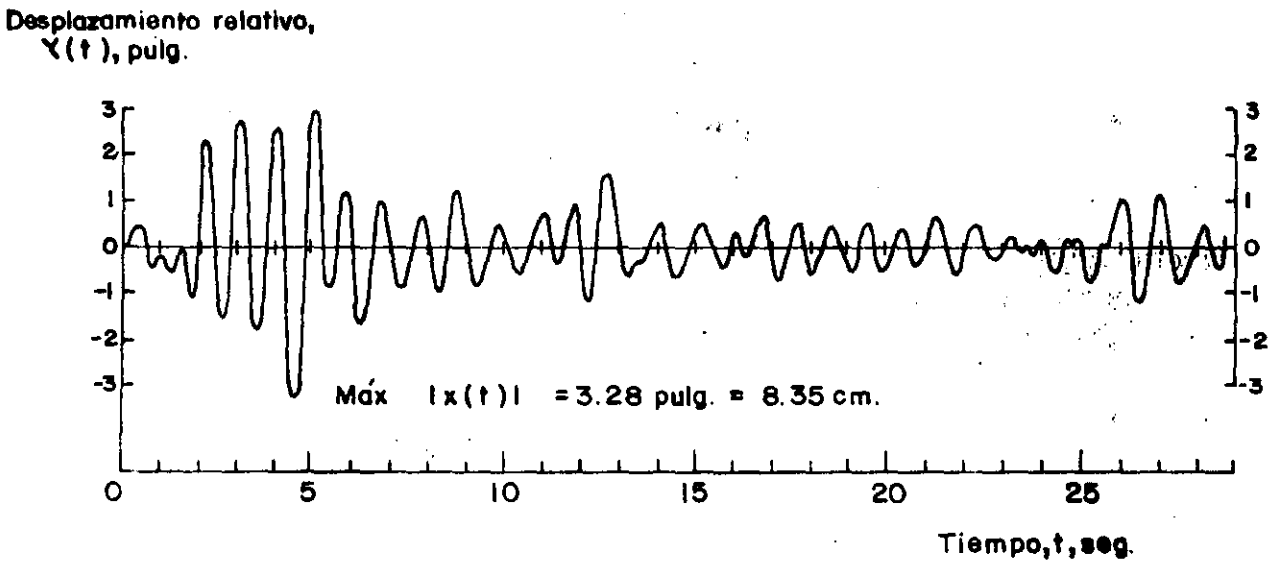
$$y_3(t) = f_3(t,\ 0.3,\ 0) = f_3(t)$$

SEAN $D_1 = \text{MAX}|y_1(t)| = D(\omega_1, \zeta)$

$$D_2 = \text{MAX}|y_2(t)| = D(\omega_2, \zeta)$$

$$D_3 = \text{MAX}|y_3(t)| = D(\omega_3, \zeta)$$

$$\vdots$$



Respuesta de un sistema amortiguado simple con $T_1 = 1.0$ seg y $\zeta' = 0.10$, al sismo de El Centro, Cal., 1940, componente N-S

1er CICLO

$$\dot{y}_{i+1} = 0 + 0.1\ (0 + 5) = 0.5 \quad ; \quad y_{i+1} = 0 + 0 + 0 + 0.008 \times 5 = 0.04$$

$$\ddot{y}_{i+1} = -1.2 \times 0.5 - 9 \times 0.04 - (-30 \times 0.2) = 5.04$$

2º CICLO

$$\dot{y}_{i+1} \doteq 0 + 0.1\ (0 + 5.04) = 0.504 \quad ; \quad y_{i+1} \doteq 0 + 0 + 0 + 0.008 \times 5.04 =$$

$$= 0.04032$$

$$\ddot{y}_{i+1} = -1.2 \times 0.504 - 9 \times 0.4032 - (-6) = 5.033 \text{ IN/SEG}^2$$

ESTOS CALCULOS SE PUEDEN ORGANIZAR MEDIANTE UNA TABLA COMO LA SIGUIENTE:

t SEG	$\ddot{x}_o$ IN/SEG^2	$\ddot{y}$ ING/SEG^2	$\dot{y}$ ING/SEG	y IN
0	0	0	0	0
0.2	-6	5.0000 5.040 5.033 5.034	0.5000 0.5040 0.5033 0.5034	0.04000 0.04032 0.04026 0.04027
0.4^-	-12	8.0000 7.442 7.534 7.533	1.8078 1.7510 1.7602 1.7601	0.26536 0.26079 0.26163 0.26162
0.4^+	0	-4.467	1.7601	0.26162
0.6	0	-6.000 -5.464 -5.550 . . .	0.7134 0.7670 0.7584 . . .	0.51204 0.51633 0.51564 . . .

EN $t = 0.2 + \Delta t = 0.4$ SEG: $\quad \dot{x}_o = -30 \times 0.4 = -12$

$$\ddot{y}_i = 5.034, \quad \dot{y}_i = 0.5034, \quad y_i = 0.04027$$

SUPONIENDO $\ddot{y}_{i+1} = 8.000$ SE OBTIENE:

1er CICLO

$$\left\{\begin{array}{l} \dot{y}_{i+1} = 0.5034 + 0.1\ (5.034 + 8.000) = 1.8068 \\ y_{i+1} = 0.04027 + 0.2 \times 0.5034 + 0.012 \times 5.034 + 0.008 \times 8 = 0.26536 \\ \ddot{y}_{i+1} = -1.2 \times 1{,}8068 - 9 \times 0.26536 - (-12) = 7.442\ \text{IN/SEG}^2 \end{array}\right.$$

EN $t = 0.4^+$ SOLO CAMBIA $\ddot{y}$: $\ddot{y}_{0.4+} = \ddot{y}_{0.4-} + \ddot{x}_o = 7.533 - 12 = -4.467$

EN $t = 0.6$, $\ddot{y}_i = -4.467)$ $\dot{y}_i = 1.7601$; $y = 0.26162$

Graphical Elements

SAPIN ..ses the Microsoft Windows graphical operating environment. There are a numbers of items specific to Windows that are described in the "Windows Users Guide", included with each version of Windows. For convenience, the basic definitions are given below. Please refer to Chapter II for instructions on using the mouse and keyboard.

Windows Version

SAPIN requires Microsoft Windows 3.0 or later and will NOT run on earlier versions. Also, SAPIN will NOT run in Windows Real mode as there is not enough memory available. This means that you MUST have an extended memory manager such as HIMEM.SYS in your CONFIG.SYS file so that Windows will run in Standard or 386 Enhanced modes. You can check the mode by clicking on "About Program Manager" under the Help menu in Windows Program Manager. Please refer to the Windows Users Guide.

Current File

This is the file name shown at the top center of the SAPIN screen. If you do the command **Save** from the **File** menu, the current structure will be saved in this file.

Current Structure

The current structure is all items that have been defined in SAPIN or read in from a file. These include joint locations, structural element definitions, structural element assignments and loads. In short, it's everything.

When you do a **Save** or **Save as** command in the **File** menu, the current structure is written into the file. When you do the command **Open** in the **File** menu, any current structure is erased and the contents of the opened file become the new current stru :.

Starting SAPIN

There are a number of ways to start SAPIN, depending on whether Windows is running or not. The suggested methods are listed below:

1. If Windows is not running, change to the drive and directory where SAPIN.EXE is located and start SAPIN from the DOS prompt by typing:

   WIN SAPIN↵

   This requires that Windows be included in the PATH statement in your AUTOEXEC.BAT file.

2. With Windows running, open the Program Manager window, click on File and then click on Run. Then enter the COMPLETE path followed by SAPIN. For example:

   E:\SAP90\SAPIN

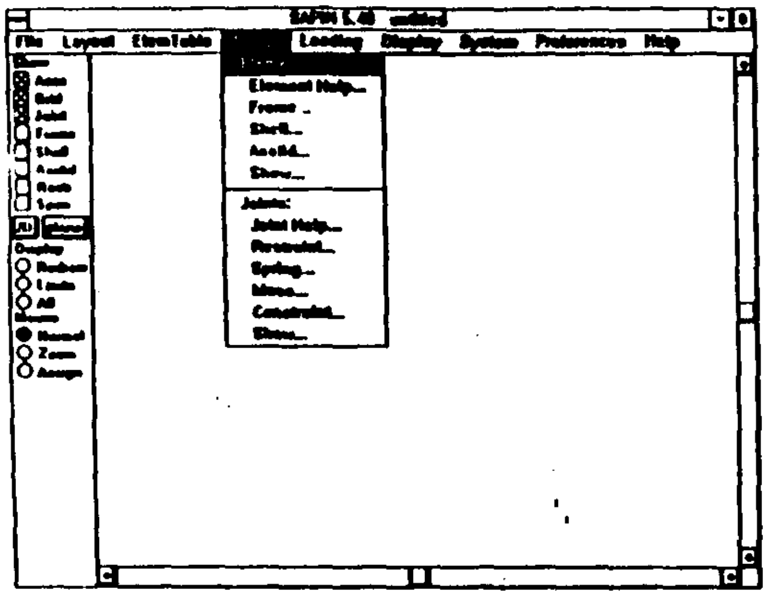
   Then click on OK to start SAI'' I.

3. With Windows running, open the File Manager and select the drive and directory containing SAPIN.EXE. Then double-click on SAPIN.EXE to start SAPIN.

4. With Windows running, it is also possible to put SAPIN.EXE in a Program Manager document icon. Refer to the Windows Users Guide for the procedure.

SAPIN Screen

When SAPIN is started, the following screen will appear:



This is the basic SAPIN screen, which is always full screen and can not be reduced in size, except as an Icon. The current file name is UNTITLED, which means that no file has yet been specified.

Menu Bar

The second line down from the top, containing **File**, **Layout**, **ElemTable**, etc. is the menu bar. Click on one of the words to bring down a menu. The **Assign** menu is shown.

Menu

The menu is the list of commands that appear when you click in the menu bar. The **Assign** menu is shown. In the menu any commands that have ... after them will bring up a dialog box. Commands without ... after them will take

immediate action. In the Assign menu, all items will bring up a dialog box.

Close Box

The small box at the upper left corner will close (stop) SAPIN if you double-click on it.

Scroll Bars

The bars at the right and bottom of the screen are the scroll bars. They have an arrow at each end. See Chapter II for the use of the scroll bars.

Control Panel

The Control Panel is at the left of the screen and contains a number of items such as Show, Display and Mouse. The Control Panel is always on screen and is a permanent dialog box with standard dialog box controls. The use of controls is discussed later in this chapter. The function of the Control Panel is to control what is shown on the screen, to switch between 2D and 3D views of the structure, to determine when the screen is redrawn, and to make it easy to assign structural elements. See Chapter IV for a description of the Control Panel functions.

TABLE of CONTENTS

I.

INTRODUCTION and TERMINOLOGY

The SAPIN program is an interactive graphical input file generator for the SAP90 finite element analysis program. SAPIN allows you to model a structure using an intuitive graphical method, while still maintaining the numerical exactness necessary for the dimensional and structural elements.

SAPIN does no analysis, but prepares and edits the input files used by program SAP90. The files are in standard ASCII text format and may be edited with any text editior if desired. See the SAP90 Users Manual "SAP90 Input Data File Structure" for a complete description of the input file format. All options of the SAP90 program are available in this release of SAPIN, except Solid elements and Heat Transfer Analysis. However, if Solid or Heat Transfer information is in the input file, SAPIN will save it and write it out unchanged.

There are a number of terms used in this manual and in the SAPIN program that are not described in the SAP90 Users Manual. These terms are described below. When you see the use of "menus", please refer to Chapter IV, which describes the menus and commands in the menus.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
Phone: (510) 845-2177
Fax: (510) 845-4096

Revised April, 1992

COMPUTERS & STRUCTURES INC.®

SAPIN™

An Interactive Graphical Input Generator for SAP90™

Ramón Cervantes Beltrán

Developed and written in U.S.A.

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
Phone: (510) 845-2177
Fax: (510) 845-4096

Revised April, 1992

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAPIN. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

SAP90™

STRUCTURAL ANALYSIS PROGRAMS

VOLUME III

COMPUTERS &
STRUCTURES
INC.

COMPUTERS & STRUCTURES INC.®

SAPIN™

An Interactive Graphical Input Generator for SAP90™

Ramón Cervantes Beltrán

Developed and written in U.S.A.

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

CURSOS ABIERTOS

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

\- 1 9 9 5 -

MOD. II ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS

SUJETAS A SISMO

C S I AN INTERACTIVE GRAPHICAL INPUT GENERATOR FOR SAP90

M. EN I. RAMON CERVANTES BELTRAN

16

Palacio de Minería Calle de Tacuba 5 Primer piso Deleg. Cuauhtémoc 06000 México, D.F. APDO. Postal M-2285
Teléfonos: 512-8955 512-5121 521-7335 521-1987 Fax 510-0573 521-4020 AL 26

REFERENCIAS.

1. Biggs, J.,M., "Introduction to Structural Dynamics", Mc Graw-Hill, 1964.

2. Hurty W. C. y Rubinstein M. F.,"Dynamics of Structures", Prentice Hall, 1964.

3. Newmark, N. M. y Rosenblueth E., "Fundamentals of Earthquake Engineering", Prentice Hall, 1971.

4. Timoshenko, S., Young, D. H. y Weaver, W. "Vibration Problems in Engineering", John Wiley, 1974.

5. Clough, R. W. y Penzien, J., "Dynamics of Structures", McGraw-Hill, 1975.

6. Craig, R. R., "Structural Dynamics: An introduction to Computer Methods", John Wiley, 1981.

7. Capra, A. y Davidovici, D., "Calcul Dynamique des Structures en zone sismique", Eyrolles, 1982.

8. Tuma, J. J. y Cheng, F. Y., "Dynamic Structural Analysis", Mc Graw-Hill Schaum's, 1983.

9. Kiseliov, V. A., "Mecánica de Construcción", Mir, 1983.

10. Paz, M., "Structural Dynamics: Theory & Computation", 2a. Ed. Van Nostrand, 1985.

11. Dowrick, D. J., "Earthquake Resistant Design: A Manual for Engineers and Architects", 2a. Ed., John Willey, 1986.

12. Anónimo, "Reglamento de Construcciones para el Distrito Federal", Junio 17, 1987; Diario Oficial de la Federación, Julio 3, 1987

13. Anónimo, "Normas Técnicas Complementarias para Diseño por Sismo", Gaceta Oficial del Departamento del Distrito Federal, Noviembre 5,1987

14. Gil et al, "Manual de Análisis Sísmico de Edificios", DDF-PNUD-Habitat, 1988.

15. Rioboó et al, "Manual para la Estructuración de Edificios", DDF-PNUD-Habitat, 1988.

16. Naeim, F., "The seismic Design Handbook", Van Nostrand, 1989.

VIBRACIONES FORZADAS SIN AMORTIGUAMIENTO.

- ECUACIONES DE EQUILIBRIO.
- RESPUESTA MODAL.
- SUPERPOSICION.

VIBRACIONES FORZADAS CON AMORTIGUAMIENTO.

- ECUACIONES DE EQUILIBRIO.
- RESPUESTA MODAL.
- SUPERPOSICION.

METODOS NUMERICOS (FRECUENCIAS Y CONFIGURACIONES MODALES)

METODO DE STODOLA-VIANELLO-NEWMARK.

- PROCEDIMIENTO.
- APLICACION AL MODO FUNDAMENTAL.
- PROCEDIMIENTO PARA MODOS SUPERIORES.
- APLICACION A MODOS SUPERIORES.
- APLICACION A ESTRUCTURAS DE FLEXION.

METODO DE HOLZER.

- PROCEDIMIENTO.
- APLICACIONES.

OTROS METODOS.

- SOLUCIONES ITERATIVAS:

  . ITERACION: INVERSA; DIRECTA; RAYLEIGH.

  . ORTOGONALIZACION GRAM-SCHMIDT.

- SOLUCIONES DE TRANSFORMACION:

  . JACOB1.

  . HOUSEHOLDER Q-R.

APLICACIONES.

ANALISIS SISMICO CONFORME RCDF93 + NTC'S

ZONIFICACION GEOTECNICA.

COEFICIENTES Y ESPECTROS PARA DISEÑO SISMICO.

FACTORES DE COMPORTAMIENTO SISMICO.

CONDICIONES DE REGULARIDAD.

CONDICIONES BASICAS DE CARGA.

- CARGAS PERMANENTES.

  . CARGAS MUERTAS.

  . CARGAS VIVAS.

- CARGAS ACCIDENTALES.

  . SISMO.

COMBINACIONES DE CARGA PARA DISEÑO.

REVISION DE ESTADOS LIMITE DE SERVICIO.

REVISION DE ESTADOS LIMITE DE FALLA.

EL PROBLEMA DEL DISEÑO SISMICO DE CONSTRUCCIONES

GENERACION DE SISMOS.

PROPAGACION / ATENUACION DE ONDAS SISMICAS.

EFECTOS LOCALES EN EL SITIO.

RESPUESTA SISMICA DE CONSTRUCCIONES.

REGLAMENTOS DE CONSTRUCCION.

CONCIENCIA SOCIAL.

EL PROCESO DE DISEÑO Y CONSTRUCCION DE ESTRUCTURAS

INVESTIGACION PRELIMINAR.

DISEÑO CONCEPTUAL.

DISEÑO PRELIMINAR.

DISEÑO FINAL.

DOCUMENTACION Y CONCURSO.

CONSTRUCCION Y SUPERVISION.

MANTENIMIENTO Y CONSERVACION.

ANALISIS SISMICO DE ESTRUCTURAS METODOS DINAMICOS Y ESTATICOS

DINAMICO INELASTICO TRIDIMENSIONAL (DOMINIO DEL TIEMPO)

DINAMICO INELASTICO PLANO (DOMINIO DEL TIEMPO)

DINAMICO ELASTICO TRIDIMENSIONAL
. (DOMINIO DEL TIEMPO)
. (ESPECTRO DE DISENO)

DINAMICO ELASTICO PLANO
. (DOMINIO DEL TIEMPO)
. (ESPECTRO DE DISENO)

ESTATICO EQUIVALENTE

SISTEMAS DE VARIOS GRADOS DE LIBERTAD

ECUACIONES DE EQUILIBRIO DINAMICO.

- FUERZAS DE INERCIA.
- FUERZAS DISIPADORAS.
- FUERZAS RESTAURADORAS.
- FUERZAS EXTERNAS.

VIBRACIONES LIBRES SIN AMORTIGUAMIENTO.

- ECUACIONES DE EQUILIBRIO.
- SOLUCION ALGEBRAICA.
- FRECUENCIAS NATURALES DE VIBRACION.
- CONFIGURACIONES NATURALES DE VIBRACION.
- PROPIEDADES DE ORTOGONALIDAD.
- RESPUESTA PARA CONDICIONES INICIALES.
- SUPERPOSICION.
- OTRAS SOLUCIONES.

VIBRACIONES LIBRES CON AMORTIGUAMIENTO.

- ECUACIONES DE EQUILIBRIO.
- RESPUESTA MODAL.
- SUPERPOSICION.

FACULTAD DE INGENIERIA U.N.A.M.

DIVISION DE EDUCACION CONTINUA

CURSO INTERNACIONAL DE INGENIERIA SISMICA

ANALISIS SISMICO DE ESTRUCTURAS
METODOS DINAMICOS Y ESTATICOS

SISTEMAS DE VARIOS GRADOS DE LIBERTAD

M. EN I. JOSE LUIS TRIGOS

Profesor, Facultad de Ingeniería, UNAM
Ingeniero Civil, Consultor
Apartado 74 - 171, México D. F. 09080
(525) 689 - 6888 FAX (525) 689 - 6639

MEXICO

6

ANALISIS SISMICO DE ESTRUCTURAS
METODOS DINAMICOS Y ESTATICOS

S I S T E M A S   D E   V A R I O S
G R A D O S   D E   L I B E R T A D

EL PROBLEMA DEL DISEÑO SISMICO DE CONSTRUCCIONES

EL PROCESO DE DISEÑO Y CONSTRUCCION DE ESTRUCTURAS

ANALISIS SISMICO DE ESTRUCTURAS METODOS DINAMICOS Y ESTATICOS

SISTEMAS DE VARIOS GRADOS DE LIBERTAD

APLICACIONES Y EJEMPLOS.

M. EN I. JOSE LUIS TRIGOS

Profesor, Facultad de Ingeniería, UNAM
Ingeniero Civil, Consultor
Apartado 74 - 171, México D. F. 09080
(525) 689 - 6888 FAX (525) 689 - 6639

MEXICO

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

CURSOS ABIERTOS

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

1995

MOD. 11 ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMO

SISTEMAS DE VARIOS GRADOS DE LIBERTAD

M. EN I. JOSE LUIS TRIGOS.

Palacio de Minería   Calle de Tacuba 5   Primer piso   Deleg. Cuauhtémoc 06000   México, D.F.   APDO. Postal M-2285
Teléfonos: 512-8955   512-5121   521-7335   521-1987   Fax 510-0573   521-4020 AL 26

XXIII CURSO INTERNACIONAL DE INGENIERIA SISMICA

ANALISIS DINAMICO DE ESTRUCTURAS SUJETAS A SISMO

BIBLIOGRAFIA

1. Blume, J.A., Newmark, N.M., y Corning, L.H., " Design of multistory reinforced concrete buildings for earthquake motions", Chicago: Portland Cement, 1961.

2. Montes, R. y Rosenblueth, E., "Cortantes y momentos sísmicos en chimeneas". Facultad de Ingeniería. División de Estudios de Posgrado, UNAM. 1968. México

3. Lainez-Lozana, Navarro, Asocs, "Comportamiento de las construcciones de adobe ante movimientos sísmicos". Elaborado por la Asociación de Asegurado del Perú. Perú, 1970.

4. Dowrick, D.J., "Earthquake resistant design: a manual for engineers and architects". J. Wiley, 1977. London.

5. Naciones Unidas, Departamento de Asuntos Económicos y Sociales, Centro de Vivienda, Construcción y Planificación. "Construcción económica resistente a sismos y huracanes". Naciones Unidas, 1976. Nueva York.

6. Green, N.B., "Earthquake resistant building design and construction". Van Nostrand Reinhold. 1978. New York.

7. Green, N.B., "Edificación, diseño y construcción sismorresistente". Versión castellana de Jesús Parra. Gili 1980. Barcelona.

8. Applied Technology Council (San Francisco, Cal). "Working draft of recommended comprehensive seismic design provisions for building". National Science Foundation. 1976. San Francisco, Cal.

9. Dowrick, D.J. "Diseño de estructuras resistentes a sismos: para ingenieros y arquitectos". Versión española Trigos, J.L., García Ferrer, C.A. Limusa, 1984. México

10. Lomnitz., Rosenblueth, E. "Seismic Risk and Engineering Decisiones". Elsevier Scientific. 1976.

11. Arnold, C., Reitherman, R., "Building configuration and seismic design". J. Wiley, 1982. New York.

12. Ambrose, J., Vergun, D., "Seismic design of buildings". J. Wiley. 1985. New York.

13. Bazán, E., Meli, R., "Manual de diseño sísmico de edificios: de acuerdo con el Reglamento de Construcciones del Distrito Federal". Limusa, 1985. México.

14. Newmark, N.M., Rosenblueth, E., "Fundamentals of Earthquake Engineering". Prentice Hall. 1971.

15. Oshiro H,F., "Construcción Antisísmica". Lima, 1972.

16. Fertis, D.G., "Dynamics and Vibration of Structures". Wiley Interscience. 1973. New York.

17. Okamoto, S., "Introduction to Earthquake Engineering". J. Wiley. 1973.

18. Newmark, N.M., Rosenblueth, E. "Fundamentos de Ingeniería Sísmica". Diana, 1976. México

19. Clough, R.W., Penzien, J. "Dynamics of Structures". McGraw Hill 1975. México.

20. Blevins, R.D., "Formulas for natural frequency and mode shape". Van Nostrand Reinhold, 1979. New York.

21. Paz, M. "Structural dynamics: Theory and computation". Van Nostrand Reinhold. 1980. New York.

22. Craig, R.R., "Structural dynamics: an introduction to computer methods". Wiley, 1981. New York.

23. Major, A., "Dynamics in civil engineering: analysis and design". Akademiai Kiado, 1980. Budapest.

XV CURSO INTERNACIONAL DE INGENIERIA SISMICA

ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS SUJETAS A SISMO

Coordinador: Dr. Octavio A. Rascón Chávez

El siguiente material bibliográfico se encuentra a su disposición en el - Centro de Información y Documentación "Ing. Bruno Mascanzoni".

PUBLICACIONES PERIODICAS

NEIL, M. "Anchorage of reinforcing bars for seismic forces". -- En: ACI - Structural journal. -- 84 (5) : p. 407-418. -- Sep./Oct. 1987.

AHMAD, J. "Earthquake resistance of reinforced concrete interior connections including a floor slab". -- En: ACI Structural journal. -- 84 (5) : p. 400-406. -- Sep./Oct. 1987.

TEGOS, I.A. "Seismic resistance of short columns and coupling beams reinforced with inclined bars". -- En: ACI Structural journal. -- 85 (1) : p. 82-86. -- Jan./Feb. 1988.

MIRZA, S.A. "Limit states design of concrete slender columns". -- En: Canadian Journal of civil engineering. -- 14 (4) : p. 439-446. -- Aug. - 1987.

RAINER, J. H. "Force reduction factors for the seismic provisions of the National Building Code of Canada". -- En: Canadian Journal of civil engineering. -- 14 (4) : p. 447-454. -- Aug. 1987.

MARCUSON, William. "Shake-proof dams". -- En: Civil engineering. -- 57 (12) : p. 44-47. -- Dec. 1987.

FANTOZZI, Mark W. "Teleport new wave engineering". -- En: Civil engineering. -- 57 (9) : p. 48-49. -- Sep. 1987.

SNYDER, Gary M. "Earthquakes will not damage this bridge". -- En: Civil - engineering. -- 57 (9) : p. 54-55. -- Sep. 1987.

MARTIN, Geoffrey. "Quake-resistant transport". -- En: Civil engineering. -- 57 (5) : p. 60-61. -- May. 1987.

CAMPANELLA, R. G. "Seismic cone penetration testing in the near offshore of the Mackenzie Delta". -- En: Canadian geotechnical journal. -- 24 (1) : p. 154-159. -- Feb. 1987.

ABDEL-GHAFFAR, Ahmed. "Elasto-plastic seismic response of 3-D earth dams : theory". -- En: Geotechnical engineering. -- 113 (11) : p. 1239-1308. -- Nov. 1987.

ELGAMAL, Ahemed-Waeil. "Elasto-plastic seismic response of 3-D earth dams : application". -- En: Geotechnical engineering. -- 113 (11) : p. 1309 -1325. -- Nov. 1987.

HANSON, R. D. "Performance of steel structures in the September 19 and 20, 1985 Mexico earthquakes". -- En: Earthquake spectra. -- 3 (2) : p. 329 -346. -- May. 1984.

SUAREZ, Luis E. "Floor response spectra with structure-equipment interaction effects by a mode synthesis approach". -- En: Earthquake engineering & structural dynamics. -- 15 (1) : p. 141-158. -- Jan. 1987.

WERNER, S. D. "Seismic response evaluation of Meloland Road Overpass Using 1979 imperial valley earthquake records". -- En: Earthquake engineering & structural dynamics. -- 15 (2) : p. 249-274. -- Feb. 1987.

LOTFI, Vahid. "A technique for the analysis of the response of dams to - earthquake". -- En: Earthquake engineering & structural dynamics. -- 15 (4) : p. 463-490. -- May. 1987.

KERR, Arnold D. "Validation of new equations for dynamic analyses of tall frame-type structures". -- En: Earthquake engineering & structural dynamics. -- 15 (5) : p. 549-563. -- Jul. 1987.

ZEMBATY, Zbigniew. "On the reliability of tower-chcaped structures under seismic excitations". -- En: Earthquake engineering & structural dynamics. -- 15 (6) : p. 761-775. -- Aug. 1987.

SING, Mahendra P. "Seismic response analysis of structure-equipment systems with non-classical damping effects". -- En: Earthquake enginee - ring & structural dynamics. -- 15 (7) : p. 871-888. -- Oct. 1987.

LA SOLUCION DE LA PRIMERA ES:

$$\theta(z) = A_1 \text{ sen } az + A_2 \cos az + A_3 \text{ senh} az + A_4 \cosh az \qquad (12)$$

EN DONDE LAS A_i SE CALCULAN EN FUNCION DE LAS CONDICIONES DE FRONTERA DE LA VIGA EN AMBOS EXTREMOS.

EJEMPLO

VIGA SIMPLEMENTE APOYADA

LAS CUATRO CONDICIONES DE FRONTERA SON:

en $z=0$: $\theta(o)=0$, $M(o)= EI\ \ddot{\theta}(o) = 0$

en $z=L$: $\theta(L)=0$, $M(L)= EI\theta''(L) = 0$

SUSTITUYENDO $\theta(o)=0$ Y $\theta''(o)=0$ EN LA EC.(12) Y SU SEGUNDA DERIVADA:

$$\left.\begin{aligned} \theta(o) &= A_2 + A_4 \cosh 0 = 0 \\ \ddot{\theta}(o) &= a^2(-A_2 + A_4 \cosh 0) = 0 \end{aligned}\right\} \Rightarrow A_2 = A_4 = 0$$

HACIENDO LO MISMO CON $\theta(L) = 0$ y $\theta''(L) = 0$:

$$\left.\begin{aligned} \theta(L) &= A_1 \text{ sen } aL + A_3 \text{ senh } aL = 0 \\ \ddot{\theta}(L) &= a^2(-A_1 \text{ sen } aL + A_3 \text{ senh } aL) = 0 \end{aligned}\right\} \rightarrow A_3 = 0$$

POR LO TANTO, $\theta(L) = A_1 \text{ sen } aL = 0$

PUESTO QUE $A_1=0$ ES LA SOLUCION TRIVIAL, SE DEBE TENER QUE A_1 SEA ARBITRARIA Y QUE

$$\text{sen } aL = 0 \rightarrow aL = n\pi; \quad n = 0, 1, 2, \ldots, \infty$$

POR LO TANTO, $a = n\pi/L$. RECORDANDO QUE

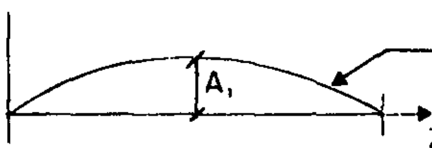
$a^4 = \omega^2\bar{m}/EI$, SE TIENE QUE

$$\omega_n^2 = (n\pi/L)^4 EI/\bar{m} \qquad O \qquad \omega_n = \frac{n^2\pi^2}{L^2}\sqrt{EI/\bar{m}}$$

SON LAS FRECUENCIAS CIRCULARES NATURALES DE VIBRACION DE LA VIGA.

LAS CONFIGURACIONES MODALES SON

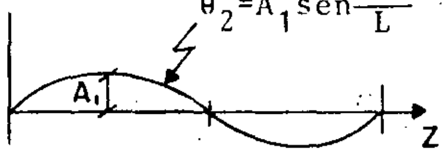
$$\theta_n(z) = A_1 \text{ sen } \frac{n\pi}{L} z$$



$\theta_1 = A_1 \text{sen } \frac{\pi z}{L}$: $\omega_1 = \frac{\pi^2}{L^2}\sqrt{EI/\bar{m}}$

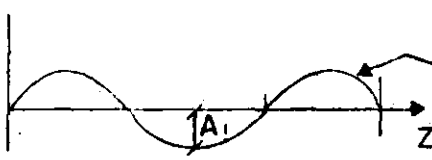
1er. MODO

$\theta_2 = A_1 \text{sen} \frac{2\pi z}{L}$



$\omega_2 = \frac{4\pi^2}{L^2}\sqrt{EI/\bar{m}}$

2o. MODO



$\theta_3 = A_1 \text{sen } \frac{3\pi z}{L}$; $\omega_3 = \frac{9\pi^2}{L^2}\sqrt{EI/\bar{m}}$

3er. MODO

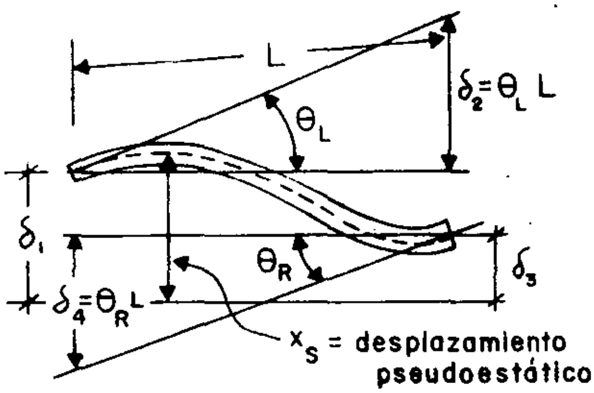
⋮

$$\omega_1 : \omega_2 : \omega_3 :: 1 : 4 : 9$$

$$\omega_i = n^2 \omega_1$$

x_s = DESPLAZAMIENTO PSEUDOESTATICO OCASIONADO POR EL MOV. DE LOS APOYOS DE MANERA ESTATICA

x = DESPLAZAMIENTO DINAMICO



SI SE TIENE UNA ROTACION Y UNA TRASLACION POR APOYO:

$$x_s = \sum_{i=1}^{4} \emptyset_i \delta_i(t) \qquad (8)$$

$\emptyset_i(z)$ = CONFIGURACION DE LA VIGA DEBIDA A $\delta_i = 1$

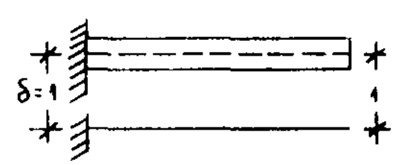
INCORPORANDO (8) EN (7):

$$P_{efect} = -\sum_{i=1}^{4} \{m\emptyset_i \ddot{\delta}_i(t) + c\,\emptyset_i \dot{\delta}_i(t) + \frac{\partial^2}{\partial z^2}\left[I(z)\frac{\partial^2 \emptyset_i(z)}{\partial z^2}(\delta_i(t)c_d + E)\right]\} \qquad (9)$$

EN LA MAYORIA DE LOS CASOS EL AMORTIGUAMIENTO INFLUYE POCO EN LA FUERZA EFECTIVA Y LA EC.(9) SE SIMPLIFICA A

$$P_{efect} = -\sum_{i=1}^{4} m\emptyset_i(z)\ddot{\delta}_i(t)$$

EN EL CASO DE UN VOLADIZO:



$$\emptyset_1(z) = 1$$

Y

$$P_{efect} = -\,m(z)\,\ddot{\delta}_1(t)$$

ANALISIS DE VIBRACIONES LIBRES

CONSIDEREMOS UNA VIGA DE SECCION CONSTANTE (EI= CONSTANTE ; $\bar{m}$=MASA POR UNIDAD DE LONGITUD).

DE LA EC.(5): $EI\,\dfrac{\partial^4 x}{\partial z^4} + \bar{m}\,\dfrac{\partial^2 x}{\partial t^2} = 0$

$$\frac{\partial^4 x}{\partial z^4} + \frac{\bar{m}}{EI}\,\frac{\partial^2 x}{\partial t^2} = 0 \qquad (10)$$

RESOLVIENDO LA EC. (10) POR SEPARACION DE VARIABLES:

$$x(z,t) = \theta(z)\,Y(t)$$

$$\theta^{IV}(z)\,Y(t) + \frac{\bar{m}}{EI}\,\theta(z)\,\ddot{Y}(t) = 0\ ;\ \frac{\theta^{IV}(z)}{\theta(z)} + \frac{\bar{m}}{EI}\,\frac{\ddot{Y}(t)}{Y(t)} = 0$$

POR LO QUE

$$\frac{\theta^{IV}(z)}{\theta(z)} = -\frac{m}{EI}\,\frac{\ddot{Y}(t)}{Y(t)} = C = a^4 \quad (\ C = \text{CONSTANTE})$$

POR LO TANTO OBTENEMOS DOS ECUACIONES DIFERENCIALES ORDINARIAS:

$$\theta^{IV}(z) - a^4\,\theta(z) = 0$$

$$\ddot{Y}(t) + \omega^2 Y(t) = 0, \quad \text{DONDE} \quad \omega^2 = \frac{a^4 EI}{\bar{m}}$$

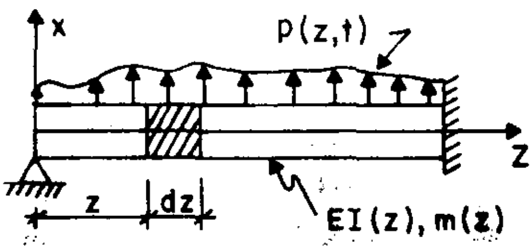
$$\text{O} \qquad a^4 = \frac{\omega^2 \bar{m}}{EI}$$

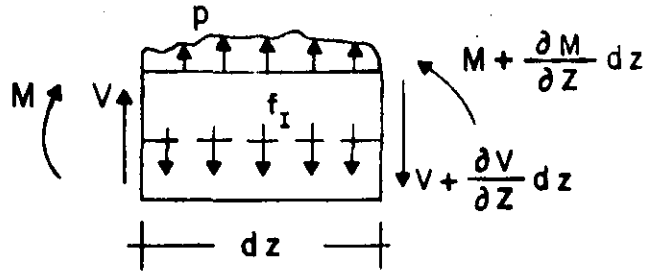
LA SOLUCION DE LA SEGUNDA DE ESTAS ES:

$$Y(t) = \frac{\dot{Y}(o)}{\omega}\,\text{sen}\,\omega t + Y(o)\cos\omega t \qquad (11)$$

VIBRACION DE VIGAS EN FLEXION

a. AMORTIGUAMIENTO NULO





$$V + pdz - (V + \frac{\partial V}{\partial z} dz) - f_I dz = 0 \tag{1}$$

EN DONDE $f_I dz = (mdz)\ \frac{\partial^2 x}{\partial t^2}$ (2)

SUSTITUYENDO (2) EN (1) Y SIMPLIFICANDO:

$$\frac{\partial V}{\partial z} = p - m\ \frac{\partial^2 x}{\partial t^2} \tag{3}$$

$$M + Vdz - (M + \frac{\partial M}{\partial z} dz) = 0\ , \qquad \frac{\partial M}{\partial z} = V \tag{4}$$

(DESPRECIANDO LOS TERMINOS DE SEGUNDO ORDEN DE LOS MOMENTOS DE p Y f_I).

SUSTITUYENDO (4) EN (3) SE OBTIENE

$$\frac{\partial^2 M}{\partial z^2} + m\frac{\partial^2 x}{\partial t^2} = p \tag{4'}$$

TOMANDO EN CUENTA QUE $\frac{M}{EI} = \frac{\partial^2 x}{\partial z^2}$ SE OBTIENE FINALMENTE

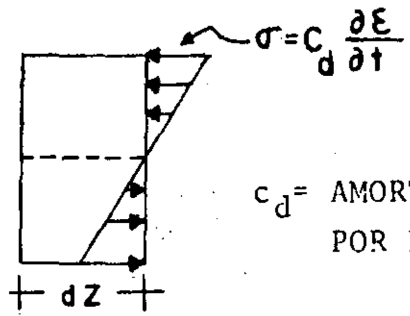
$$\frac{\partial^2}{\partial z^2}(EI\ \frac{\partial^2 x}{\partial z^2}) + m\frac{\partial^2 x}{\partial t^2} = p \tag{5}$$

b. AMORTIGUAMIENTO VISCOSO

- FUERZA DE AMORTIGUAMIENTO POR VELOCIDAD TRANSVERSAL $= c(z) \frac{\partial x}{\partial t} dz$

$$\frac{\partial V}{\partial z} = p - m \frac{\partial^2 x}{\partial t^2} - c \frac{\partial x}{\partial t} \qquad (6)$$

- FUERZA DE AMORTIGUAMIENTO POR DEFORMACION DE LA VIGA. ACEPTANDO LA HIPOTESIS DE NAVIER DE DEFORMACION PLANA



$$M_{amort} = \int \sigma y da = c_d I(z) \frac{\partial^3 x}{\partial z^2 \partial t}$$

c_d = AMORTIGUAMIENTO POR DEFORMACION

INCORPORANDO EL MOMENTO DEBIDO AL AMORTIGUAMIENTO EN LA EC. (5)

$$\frac{\partial^2}{\partial z^2} \left(EI \frac{\partial^2 x}{\partial z^2} + C_d I \frac{\partial^3 x}{\partial z^2 \partial t}\right) + m\frac{\partial^2 x}{\partial t^2} + c\frac{\partial x}{\partial t} = p \qquad (6)$$

SI LA EXCITACION ES POR MOVIMIENTO DE LOS APOYOS, SE PUEDE DEMOSTRAR (CLOUGH Y PENZIEN, PAG 303) QUE:

$$\frac{\partial^2}{\partial z^2} \left(EI \frac{\partial^2 x}{\partial z^2} + C_d I \frac{\partial^3 x}{\partial z^2 \partial t}\right) + m\frac{\partial^2 x}{\partial t^2} + c\frac{\partial x}{\partial t} = p_{efect.}$$

EN DONDE

$$p_{efect} = \frac{-\partial^2}{\partial z^2} \left(EI \frac{\partial^2 x_s}{\partial z^2} + C_d I \frac{\partial^3 x_s}{\partial z^2 \partial t}\right) - m\frac{\partial^2 x_s}{\partial t^2} - c \frac{\partial x_s}{\partial t} \qquad (7)$$

$x_{tot}(z,t) = x_s(z,t) + x(z,t)$

VIBRACIONES FORZADAS EN VIGAS DE CORTANTE

SEA $\ddot{x}_o(t)$ LA EXCITACION DEL TERRENO. LA RESPUESTA, $x(t)$, DEL SISTEMA ES

$$(3)\qquad x(t) = -\sum_{n=1}^{\infty} \frac{a_n}{\omega_n} \operatorname{sen} \frac{\omega_n}{v} X \int_0^t \ddot{x}_o(\tau) \operatorname{sen}\omega_n(t-\tau)\,d\tau$$

DONDE

$$(4)\qquad a_n = \frac{\int_0^L n \operatorname{sen} \frac{\omega_n v}{X}\,dx}{\int_0^L n \operatorname{sen}^2 \frac{\omega_n v}{X}\,dx} = \frac{4}{(2n-1)\pi}$$

TAREA: DEMOSTRAR ECS (3) Y (4) Y ESTUDIAR SECCION 3.15.

EJEMPLO: CALCULAR EL LIMITE SUPERIOR DEL CORTANTE EN UNA VIGA DE CORTANTE A CUYA BASE SE LE SOMETE A UNA ACELERACION CONSTANTE, a .

EL ESPECTRO DE ESTA EXCITACION ES $V = a/\omega$

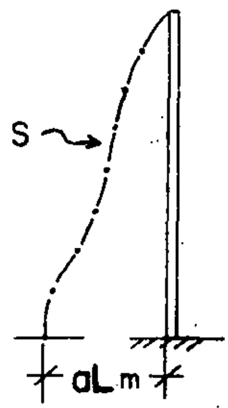
POR LO TANTO, $S \leq k \left| \frac{\partial}{\partial X} \left(\sum_{n=1}^{\infty} \frac{a_n}{\omega_n} \operatorname{sen} \frac{\omega_n}{v} X \right) \right| V$

$$S \leq \left| \sum_{n=1}^{\infty} \frac{k a_n V}{\omega_n} \frac{\omega_n}{v} \cos \frac{\omega_n}{v} X \right| = \frac{4k}{\pi}\frac{a}{v} \left| \sum_{n=1}^{\infty} \frac{\cos \frac{\pi}{2L}(2n-1)X}{(2n-1)\frac{v}{L}\,\frac{\pi}{2}(2n-1)} \right| ; \quad \text{con } v^2 = \frac{k}{m} :$$

$$S \leq \frac{8aLm}{\pi^2} \sum_{n=1}^{\infty} \frac{1}{(2n-1)^2} \left| \cos \frac{(2n-1)\pi X}{2L} \right|$$

EN X 0:

$$S \le (8\underline{aLm})/\pi^2 \underbrace{\sum_{n=1}^{\infty} \frac{1}{(2n-1)^2}}_{\pi^2/8} = aLm$$



EN EL EXTREMO X = L SE TENDRA

$$(4) \quad x(L,t) = 0 \quad \Rightarrow \frac{\omega_n L}{v} = n\pi \quad ; \quad n = 1,2,\ldots$$

PUESTO QUE EN LA EC (3) SE TOMA j=0, YA QUE j=1,2,... DAN LA MISMA SOLUCION, LO CUAL CONDUCE A $a_n = 0$.

DE LA EC (4): $\omega_n = \frac{n\pi v}{L}$; $n = 1,2,\ldots$

FRECUENCIA FUNDAMENTAL

SI $n=1$, $\quad \omega_1 = \frac{\pi v}{L} \quad \therefore \quad \omega_n = n\,\omega_1$

Y $\quad T_1 = \frac{2L}{v} \quad\quad T_n = \frac{T_1}{n}$

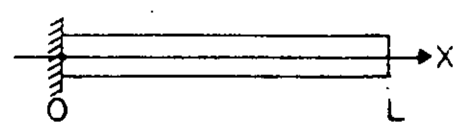
LAS CONFIGURACIONES MODALES QUEDAN:

$$Z_n = A_n \operatorname{sen}\frac{n\pi X}{L} \quad ; \quad x(t,X) = \bar{A}_n \operatorname{sen}\frac{n\pi X}{L}\operatorname{sen}\frac{n\pi v}{L}(t-t_n)$$

CONDICION DE ORTOGONALIDAD:

$$\int_0^L A_i \operatorname{sen}\frac{i\pi X}{L} A_j \operatorname{sen}\frac{j\pi X}{L}dx = 0, \quad \text{SI} \quad i \neq j$$

<u>EJEMPLO 2</u>: VIGA DE CORTANTE APOYADA EN X = 0 Y LIBRE EN X = L.



DE $x(0,t) = 0 \Rightarrow a_n = 0$

DE $x'(L,t) = 0$ (PUESTO QUE EN X = L SE DEBE CUMPLIR QUE LA FUERZA CORTANTE, S, SEA NULA),

$$x'(\underline{X},t) = A_n \frac{\omega_n}{v} \cos\frac{\omega_n X}{v} \operatorname{sen}\omega_n(t-t_n)$$

$$\therefore \quad x'(L,t) = 0 = \cos\frac{\omega_n L}{v} \Rightarrow \frac{\omega_n L}{v} = \frac{\pi}{2}(2n-1)$$

$$0 \qquad \omega_n = \frac{v}{L}\ \frac{\pi}{2}\ (2n-1) \qquad\qquad n = 1,2,\ldots$$

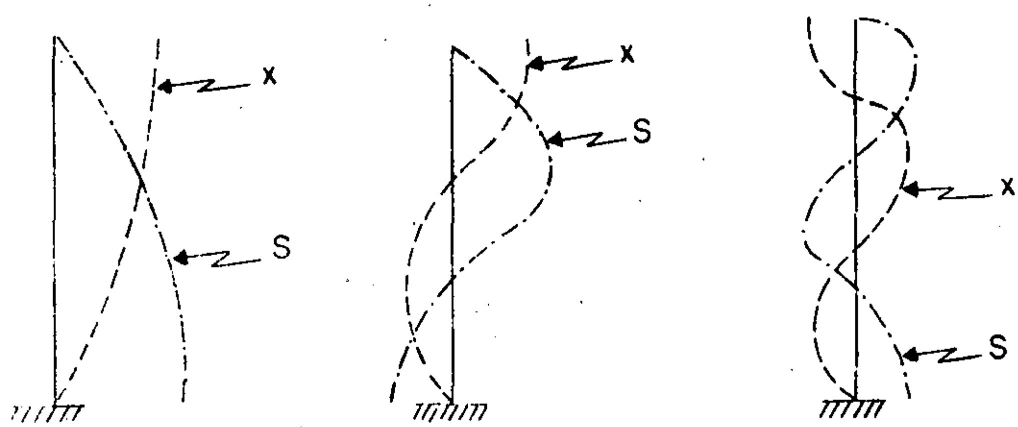
$$\text{SI} \quad n=1,\ \omega_1 = \frac{\pi v}{L.2} \Rightarrow T_1 = \frac{4L}{v}$$

$$\therefore \quad \omega_n = \omega_1(2n-1) \ ; \quad T_n = \frac{T_1}{2n-1}$$

$$\text{ASI: } T_2 = \frac{T_1}{3}\ ,\ T_3 = \frac{T_1}{5}\ ,\ \text{ETC.}$$

DISTRIBUCION DE CORTANTES:

$$S_n = k\,\frac{\partial x}{\partial X} = \overline{A}_n\ k\ \frac{\omega_n}{v}\cos\frac{\omega_n X}{v}\ \text{sen}\,\omega_n(t-t_n)$$

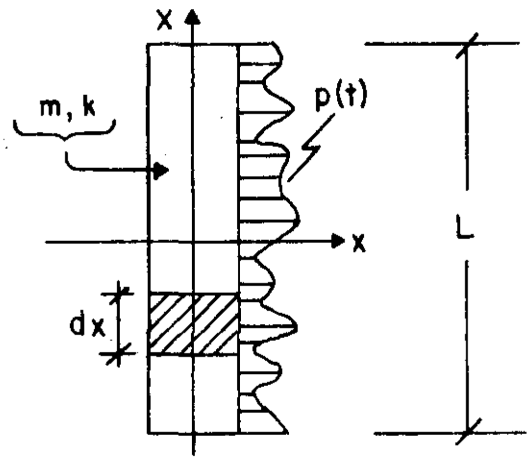


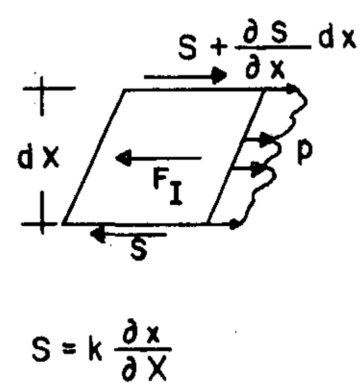
1er. MODO(FUNDAMENTAL) 2o. MODO 3er. MODO

VIGAS DE CORTANTE NO AMORTIGUADAS

SON SISTEMAS CONTINUOS CUYOS CAMBIOS DE PENDIENTE SON PROPORCIONALES AL CORTANTE QUE ACTUA EN LA SECCION.

SEAN m y p LA MASA Y FUERZA EXTERNA DISTRIBUIDAS POR UNIDAD DE LONGITUD, Y SEA k LA RIGIDEZ POR CORTANTE:





$k = FAG$

F = FACTOR DE FORMA

A = AREA SECCION TRANSVERSAL

G = MODULO DE ELASTICIDAD DINAMICO AL CORTANTE

$$F_I = (mdX) \frac{\partial^2 x}{\partial t^2}$$

POR EQUILIBRIO:

$$\frac{\partial S}{\partial X} dX + pdX - m \frac{\partial^2 x}{\partial t^2} dX = 0$$

$$m \frac{\partial^2 x}{\partial t^2} - k \frac{\partial^2 x}{\partial X^2} = p(t) \qquad (1)$$

LA EC HOMOGENEA QUEDA (CON $p=0$)

$$(2) \quad \frac{\partial^2 x}{\partial t^2} - v^2 \frac{\partial^2 x}{\partial X^2} = 0 \ ; \quad v^2 = \frac{k}{m}$$

ESCRIBIENDO $x(t) = Z_n(X)\theta_n(t)$, LA EC (2) QUEDA

$$Z_n \ddot{\theta}_n - v^2 Z_n'' \theta_n = 0$$

$$\frac{\ddot{\theta}_n(t)}{\theta_n(t)} - v^2 \frac{Z_n''}{Z_n} = 0 \Rightarrow \frac{\ddot{\theta}_n(t)}{\theta_n(t)} = v^2 \frac{Z_n''}{Z_n} = -\omega_n^2 = \text{CONSTANTE}$$

$$\Rightarrow \ddot{\theta}_n + \omega_n^2 \theta_n = 0 \quad ; \quad Z_n'' + \frac{\omega_n^2}{v^2} Z_n = 0$$

$$\theta_n = B_n \operatorname{sen} \omega_n (t-t_n), \quad Z_n = A_n \operatorname{sen} \frac{\omega_n}{v} (X - a_n)$$

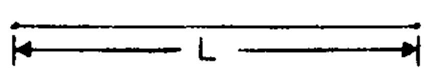
$$\therefore \ x_n = \bar{A}_n \operatorname{sen}\left[\frac{\omega_n}{v}(X-a_n)\right] \operatorname{sen}\left[\omega_n(t-t_n)\right], \quad n=1,2\ldots ; \ \bar{A}_n = B_n A_n$$

LAS CONSTANTES a_n Y ω_n SE DETERMINAN EN CADA PROBLEMA EN FUNCION DE LAS CONDICIONES DE FRONTERA.

CONDICION DE ORTOGONALIDAD:

$$\int_0^L x_n(X) x_j(X) = 0, \ \text{SI } n \neq j$$

<u>EJEMPLO 1</u>: CUERDA VIBRANTE DE LONGITUD L Y EXTREMOS FIJOS:



EN EL EXTREMO $X=0$ SE TENDRA

$$(3) \quad x(0,t) = 0 \quad \Rightarrow \frac{\omega_n a_n}{v} = j\pi \ ; \quad j = 0,1,2,\ldots \Rightarrow a_n = 0$$

Finalmente, las respuestas máximas dinámicas de la estructura en cuestión se pueden obtener mediante el método exacto haciendo uso de las ecuaciones

$$V_1 = \text{Máx}\left|\left\{ \sum_{n=1}^{2} C_n\, a_n(t) \left[m_1\, z_{1,n} + m_2\, z_{2,n} \right] \right\}\right| \tag{A.38}$$

$$V_2 = \text{Máx}\left| \sum_{n=1}^{2} C_n\, a_n(t)\, m_2\, z_{2,n} \right| \tag{A.39}$$

REFERENCIAS

1. E. Rosenblueth, **"A Basis for Aseismic Design"**, Tesis doctoral, *Universidad de Illinois,* Urbana (1951)

2. E. Rosenblueth, **"Sobre la respuesta sísmica de estructuras de comportamiento lineal"**, *Segundo Congreso Nacional de Ingeniería Sísmica,* Veracruz (1968)

3. **"Reglamento de Construcciones para el Distrito Federal"**, *Diario Oficial,* México, D. F. (feb 1966)

4. **"Los Angeles City Building Code"**, Los Angeles, Cal. (1966)

5. R. Husid, **"Estimación de la respuesta máxima de tranques de tierra sometidos a la acción de terremotos"**, *Tercer Congreso Nacional de Ingeniería Sísmica,* Acapulco (1971)

6. O. A. Rascón, **"Modelo estocástico para simular registros de temblores en terreno duro"**, Tesis doctoral, *Facultad de Ingeniería, UNAM,* México, D. F. (1968)

7. M. Chávez, O. A. Rascón y L. Alonso, **"Un nuevo método para corrección de la línea base de registros de temblores"**, *Tercer Congreso Nacional de Ingeniería Sísmica,* Acapulco (1971)

8. J. Elorduy, y E. Rosenblueth, **"Torsiones sísmicas en edificios de un piso"**, Informe 164, *Instituto de Ingeniería, UNAM,* México, D. F. (1968)

9. O. A. Rascón, **"Efectos sísmicos en estructuras en forma de péndulo invertido"**, *Revista de la Sociedad Mexicana de Ingeniería Sísmica,* Vol 3, No 1, México, D. F. (1965), pp 8-16

10. E. Rosenblueth, y L. Esteva, **"Folleto complementario: diseño sísmico de edificios, proyecto de Reglamento de Construcciones en el Distrito Federal"**, *Ediciones Ingeniería,* México, D. F. (1962)

11. T. Naylor et al, **"Técnicas de simulación en computadoras"**, *Limusa-Wiley,* México, D. F. (1971)

12. J. Hammersley y D. Handscomb, **"Monte-Carlo Methods"**, *Methuen,* Londres (1964)

Instituto de Ingeniería

División de Investigación de la Facultad de Ingeniería

Universidad Nacional Autónoma de México
Ciudad Universitaria, México 20, D. F.
MEXICO

K_t/J_c Ω^2 cuadrado de la frecuencia circular natural por rotación

$\lambda = \omega^2/p^2$

$\eta_c = \Omega^2/p^2$

se llega a

$$\gamma_{1,2} = 2\,(1 + \eta_c \pm \sqrt{(1+\eta_c)^2 - \eta_c}) \qquad \text{(A.17)}$$

Por otra parte, los vectores de las configuraciones modales son

$$\begin{bmatrix} x_n \\ \\ \\ \Phi_n \end{bmatrix} = \begin{bmatrix} 1 \\ \\ \\ \dfrac{4-\lambda_n}{2L} \end{bmatrix} ;\ n = 1, 2 \qquad \text{(A.18)}$$

Se puede demostrar (ref 9) que los coeficientes de participación correspondientes a los modos 1 y 2 se encuentran dados por la expresión

$$C_n = \frac{x_n\, m}{x_n^2\, m + \Phi_n^2\, J_c};\ n = 1, 2 \qquad \text{(A.19)}$$

Partiendo del hecho de que se conocen las aceleraciones espectrales de cada modo, a_n, la fuerza cortante máxima y el momento máximo de cabeceo correspondientes valen

$$V_n = m\, a_n\, C_n\, x_n = m\, a_n\, C_n ;\ n = 1, 2 \qquad \text{(A.20)}$$

$$M_n = J\, a_n\, C_n\, \Phi_n = J\, a_n\, C_n\, \frac{4-\lambda_n}{2L} =$$

$$= \frac{(4-\lambda_n)\, J_c}{2\, Lm}\, V_n \qquad \text{(A.21)}$$

Las respuestas dinámicas de la estructura de acuerdo con los criterios del Reglamento de Construcciones del D. F. (método 1) y de Rosenblueth (método 2), se obtienen haciendo uso de las ecuaciones

$$\hat{V} = \sqrt{V_1^2 + V_2^2} \qquad \text{(A.22)}$$

$$\hat{M} = \sqrt{M_1^2 + M_2^2} \qquad \text{(A.23)}$$

$$\tilde{V} = \sqrt{V_1^2 + V_2^2 + 2\,\frac{V_1\, V_2}{1+\epsilon_{12}^2}} \qquad \text{(A.24)}$$

$$\tilde{M} = \sqrt{M_1^2 + M_2^2 - 2\,\frac{M_1\, M_2}{1+\epsilon_{12}^2}} \qquad \text{(A.25)}$$

donde ϵ_{12}^2 se calcula mediante la ec 1.3. El signo menos aparece en la ec A.25 debido a que la función de transferencia del segundo modo es de signo opuesto a la del primero, ya que se puede demostrar, a partir de la ec A.17, que $\lambda_1 \leqslant 4$ y $\lambda_2 \geqslant 4$, por lo que el factor $4 - \lambda_n$ que aparece en la ec A.21 tiene signo positivo en el modo 1, y negativo en el 2.

La respuesta dinámica exacta se obtiene utilizando las expresiones

$$V = \text{Máx}\ \left|\left\{ C_1\, m\, x_1\, a_1(t) + C_2\, m\, x_2\, a_2(t) \right\}\right| \qquad \text{(A.26)}$$

$$M = \text{Máx}\ \left|\left\{ C_1\, J_c\, \Phi_1\, a_1(t) + C_2\, J_c\, \Phi_2\, a_2(t) \right\}\right| \qquad \text{(A.27)}$$

A.3 ANALISIS DINAMICO DE UNA ESTRUCTURA SUJETA A TRASLACION

Consideremos ahora el caso de una estructura de cortante de dos pisos, en la cual no existe rotación en los planos horizontales en los niveles de los pisos (fig A.4).

La ecuación matricial de equilibrio de este sistema es (ref 10)

$$\begin{bmatrix} m_1\,\omega^2 - K_1 - K_2 & K_2 \\ \\ K_2 & m_2\,\omega^2 - K_2 \end{bmatrix} \begin{bmatrix} z_1 \\ \\ z_2 \end{bmatrix} = \{0\} \qquad \text{(A.28)}$$

donde m_1 y m_2 son las masas concentradas en los niveles 1 y 2, y K_1 y K_2 son las rigideces de los entrepisos 1 y 2, respectivamente.

Partiendo de este sistema de ecuaciones y haciendo $\eta_t = (K_2/m_2)/(K_1/m_1)$ y $\lambda = \omega^2/(K_1/m_1)$, se obtienen las raíces

$$\lambda_{1,2}^2 = \frac{1}{2}\left[\eta_t + (m_2/m_1)\,\eta_t + 1\right] \pm$$

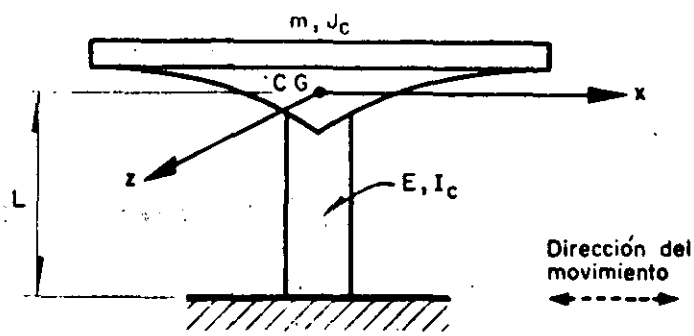


Fig A.2. Estructura en forma de péndulo invertido (vista lateral)

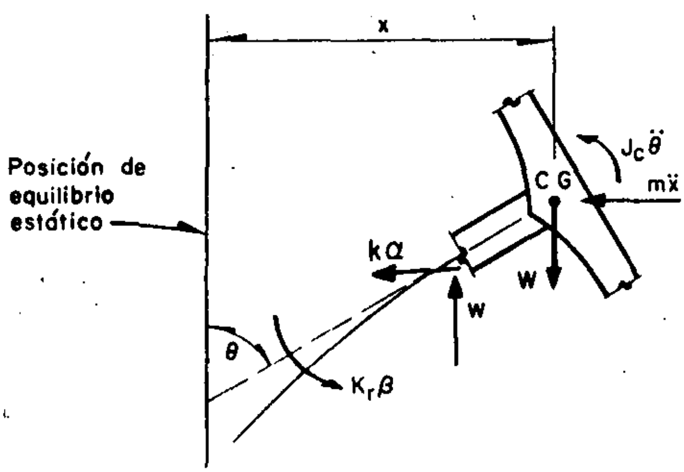


Fig A.3. Diagrama de cuerpo libre de la estructura de la fig A.2

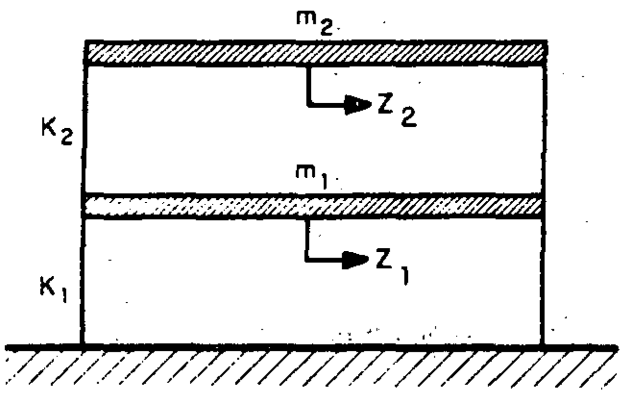


Fig A.4. Estructura de dos pisos sujeta a traslación (vista lateral)

$$\pm\frac{1}{2}\sqrt{\left[\eta_t + (m_2/m_1)\,\eta_t + 1\right]^2 - 4\,\eta_t} \qquad \text{(A.29)}$$

Además, los vectores de configuraciones modales resultan ser

$$\begin{bmatrix} z_{1,n} \\ \\ z_{2,n} \end{bmatrix} = \begin{bmatrix} 1 \\ \\ 1 + \dfrac{(K_1/m_1) - \omega_n^2}{\eta_t\,(K_1/m_1)(m_2/m_1)} \end{bmatrix};\ n = 1, 2 \qquad \text{(A.30)}$$

Además, se puede demostrar (ref 10) que los coeficientes de participación de los modos 1 y 2 se encuentran dados por

$$C_n = \frac{z_{1,n} + (m_2/m_1)\,z_{2,n}}{z_{1,n}^2 + (m_2/m_1)\,z_{2,n}^2};\ n = 1, 2 \qquad \text{(A.31)}$$

Si se conocen las aceleraciones espectrales de cada modo, a_n, la fuerza cortante máxima correspondiente al entrepiso 1 en cada modo vale

$$V_{1,n} = C_n\,a_n\,(m_1\,z_{1,n} + m_2\,z_{2,n})\ ;\ n = 1, 2 \qquad \text{(A.32)}$$

en tanto que la fuerza cortante máxima correspondiente al entrepiso 2 es

$$V_{2,n} = C_n\,a_n\,m_2\,z_{2,n}\ ;\ n = 1, 2 \qquad \text{(A.33)}$$

Ya conocidos los valores de $V_{1,n}$ y $V_{2,n}$, las respuestas máximas dinámicas totales de la estructura estimadas con los métodos 1 y 2 se calculan haciendo uso de las fórmulas

$$\hat{V}_1 = \sqrt{V_{1,1}^2 + V_{1,2}^2} \qquad \text{(A.34)}$$

$$\hat{V}_2 = \sqrt{V_{2,1}^2 + V_{2,2}^2} \qquad \text{(A.35)}$$

$$\tilde{V}_1 = \sqrt{V_{1,1}^2 + V_{1,2}^2 + 2\,\frac{V_{1,1}\,V_{1,2}}{1 + \epsilon_{12}^2}} \qquad \text{(A.36)}$$

$$\tilde{V}_2 = \sqrt{V_{2,1}^2 + V_{2,2}^2 - 2\,\frac{V_{2,1}\,V_{2,2}}{1 + \epsilon_{12}^2}} \qquad (\quad)$$

donde ϵ_{12}^2 se calcula mediante la ec 1.3

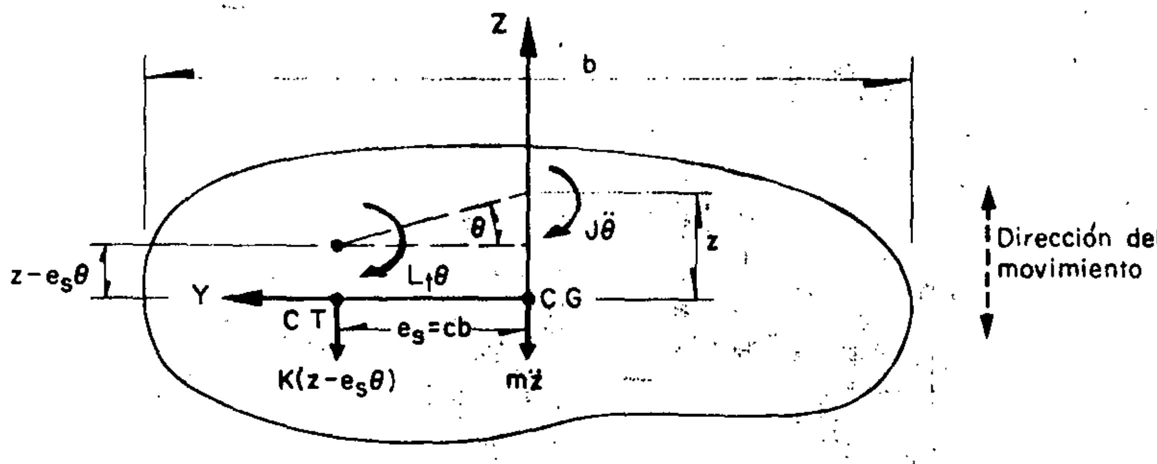


Fig A.1. Diagrama de cuerpo libre de una estructura sujeta a torsión y traslación (vista superior)

$$\begin{bmatrix} z_n \\ \\ \Phi_n \end{bmatrix} = \begin{bmatrix} 1 \\ \\ \dfrac{1-\lambda_n^2}{cb} \end{bmatrix} \; ; n = 1, 2 \qquad (A.4)$$

En términos de las raíces $\lambda_{1,2}^2$ de la ec A.3, se puede demostrar (ref 8) que los coeficientes de participación de los modos 1 y 2 (las proporciones en que contribuyen los modos a la respuesta total del sistema) se encuentran dados por

$$C_n = \frac{c^2}{c^2 + (1-\lambda_n^2)^2\, j^2} \; ; n = 1, 2 \qquad (A.5)$$

Ahora, si se suponen conocidas las aceleraciones espectrales de cada modo, a_n, la fuerza cortante máxima vale

$$V_n = m\, a_n\, C_n \; ; n = 1, 2 \qquad (A.6)$$

y el momento torsionante máximo respecto al centro de torsión es

$$M_n \; \frac{(1-\lambda_n^2)\, J\, V_n}{cbm} \; ; n = 1, 2 \qquad (A.7)$$

Una vez conocidos los valores de V_1, V_2, M_1 y M_2, la aplicación de la ec 1.2 conduce a la estimación de la fuerza cortante y del momento torsionante máximos mediante el método 2; ellos son, respectivamente

$$\tilde{V} = \sqrt{V_1^2 + V_2^2 + 2\,\frac{V_1\, V_2}{1+\epsilon_{12}^2}} \qquad (A.8)$$

$$\tilde{M} = \sqrt{M_1^2 + M_1^2 - 2\,\frac{M_1\, M_2}{1+\epsilon_{12}^2}} \qquad (A.9)$$

donde ϵ_{ij}^2 se obtiene aplicando la ec 1.3. El signo negativo asociado al doble producto que aparece en la ec A.9 se debe a que las funciones de transferencia de los momentos en el primero y segundo modos tienen signo contrario, ya que el factor $(1 - \lambda_n^2)$ que aparece en la ec A.7 es positivo para el primer modo ($n = 1$) y negativo para el segundo ($n = 2$), lo cual se demuestra como sigue:

De la ec A.3

$$\lambda_1^2 = \frac{\eta+1}{2} - \sqrt{\frac{(\eta-1)^2}{4} + \frac{c^2}{j^2}}$$

por lo que

$$\lambda_1^2 \leqslant \frac{\eta+1}{2} - \frac{\eta-1}{2} = 1$$

Análogamente

$$\lambda_2^2 = \frac{\eta+1}{2} + \sqrt{\frac{(\eta-1)^2}{4} + \frac{c^2}{j^2}}$$

de ahí que, si $\eta \geqslant 1$

$$\lambda_2^2 \geq \frac{\eta+1}{2} + \frac{\eta-1}{2} = \eta$$

o, si $\eta \leq 1$

$$\lambda_2^2 \geq \frac{\eta+1}{2} + \frac{1-\eta}{2} = 1$$

En consecuencia, $(1 - \lambda_1^2) \geq 0$ y $(1 - \lambda_2^2) \leq 0$

Además, según el Reglamento del Distrito Federal (método 1) las respuestas dinámicas máximas del mismo sistema estarían dadas por (ec 1.1)

$$\hat{V} = \sqrt{V_1^2 + V_2^2} \qquad (A.10)$$

$$\hat{M} = \sqrt{M_1^2 + M_2^2} \qquad (A.11)$$

Finalmente, por el método exacto, las respuestas máximas totales, V y M, se obtienen localizando los máximos en el tiempo, t, de las sumas de las respuestas (cortante o momento, según sea el caso) en los modos 1 y 2, es decir,

$$V = \text{Máx} \left| \{ m\, C_1\, a_1\, (t) + m\, C_2\, a_2 (t) \} \right| =$$

$$= \text{Máx} \left| \{ V_1 (t) + V_2 (t) \} \right| \qquad (A.12)$$

$$M = \text{Máx} \left| \{ \Gamma_1\, V_1 (t) + \Gamma_2\, V_2 (t) \} \right| \qquad (A.13)$$

donde

$$\Gamma_n = \frac{(1 - \lambda_n^2)\, J}{cbm} \; ; \; n = 1, 2 \qquad (A.14)$$

A.2 ANALISIS DINAMICO DE UNA ESTRUCTURA SUJETA A CABECEO

Es frecuente que en la práctica se presenten estructuras constituidas por una hilera de columnas o una sola columna que sostiene una losa o un cascarón (péndulos invertidos), tal como la que aparece en la fig A.2. La respuesta dinámica de una estructura de este tipo se debe obtener (ref 9) considerando el efecto que la inercia rotacional de la cubierta induce en el movimiento total del sistema.

En la fig A.2 se empleó la notación:

- W peso de la cubierta más la parte tributaria de columna
- m W/g
- g aceleración de la gravedad
- J_c momento de inercia de la masa de la cubierta respecto al eje Z
- E módulo de elasticidad del material de la columna
- I_c momento de inercia de la sección transversal de la columna respecto al eje Z
- CG centro de gravedad de la cubierta
- L distancia del suelo al centro de gravedad

El diagrama de cuerpo libre de la estructura anterior aparece en la fig A.3, en la cual se tiene que (ref 9)

- K rigidez por traslación $= 3EI_c/L^3$
- K_r rigidez por rotación $= EI_c/L$
- x desplazamiento del centro de gravedad de la cubierta
- Φ rotación del centro de gravedad de la cubierta

$\alpha = (x - k_r\, \gamma\, \phi)/k$

$\beta = (\phi - k\, \gamma\, x)/k$

$\gamma = L^2/2\, EI_c$

$k = 1 - K\, L^3/4EI_c = 0.25$

Las ecuaciones diferenciales de movimiento correspondientes al diagrama de cuerpo libre de la estructura son

$$\begin{aligned} m\, \ddot{x} + (Kx - K\, K_r\, \gamma\, \Phi)/k &= 0 \\ J_c\, \ddot{\Phi} + (K_r\, \Phi - K\, K_r\, \gamma x)/k &= 0 \end{aligned} \qquad (A.15)$$

Considerando que se satisfacen las relaciones $\ddot{x} = -\omega^2 x$ y $\ddot{\theta} = -\omega^2 \theta$, donde ω es la frecuencia circular natural de vibración de la estructura, y resolviendo el sistema de ecuaciones A.15, se obtiene la ecuación característica

$$\omega^4 - \frac{K\, J_c + m\, K_r}{m\, J_c\, k}\, \omega^2 + \frac{K\, K_r}{4m\, J_c\, k^2} = 0 \qquad (A.16)$$

que es una ecuación de segundo grado en ω^2. Si se efectúan algunas transformaciones algebraicas y se considera que

$K/m = p^2$ cuadrado de la frecuencia circular natural por traslación

Los promedios globales de las fuerzas cortantes normalizadas fueron, para $\zeta = 0$, 1.15; para $\zeta = 0.05$, 1.04, y en cuanto a $\zeta = 0.10$, 1.04. Además, se observa que respecto a $\zeta = 0.05$ y 0.10, los resultados son muy similares, es decir, son independientes de ζ si $\zeta \geq 0.05$

— Las medias de los resultados normalizadas son estadísticamente independientes de la relación de masas, m_2/m_1, con un nivel de confianza de 95 por ciento, pero los casos especiales de m_2/m_1 y η_t indicados anteriormente tuvieron mayor dispersión

Para $\zeta = 0.05$ y 0.10, las estimaciones normalizadas son estadísticamente independientes de η_t, con un nivel de confianza de 95 por ciento, como puede apreciarse en la fig 24 en la que aparecen únicamente los resultados del método 2. Para $\zeta = 0$ esta hipótesis no se aceptó

— Por la misma razón indicada en el último párrafo de conclusiones del problema de cabeceo, las estimaciones obtenidas con ambos métodos son en promedio satisfactorias en este tipo de estructuras

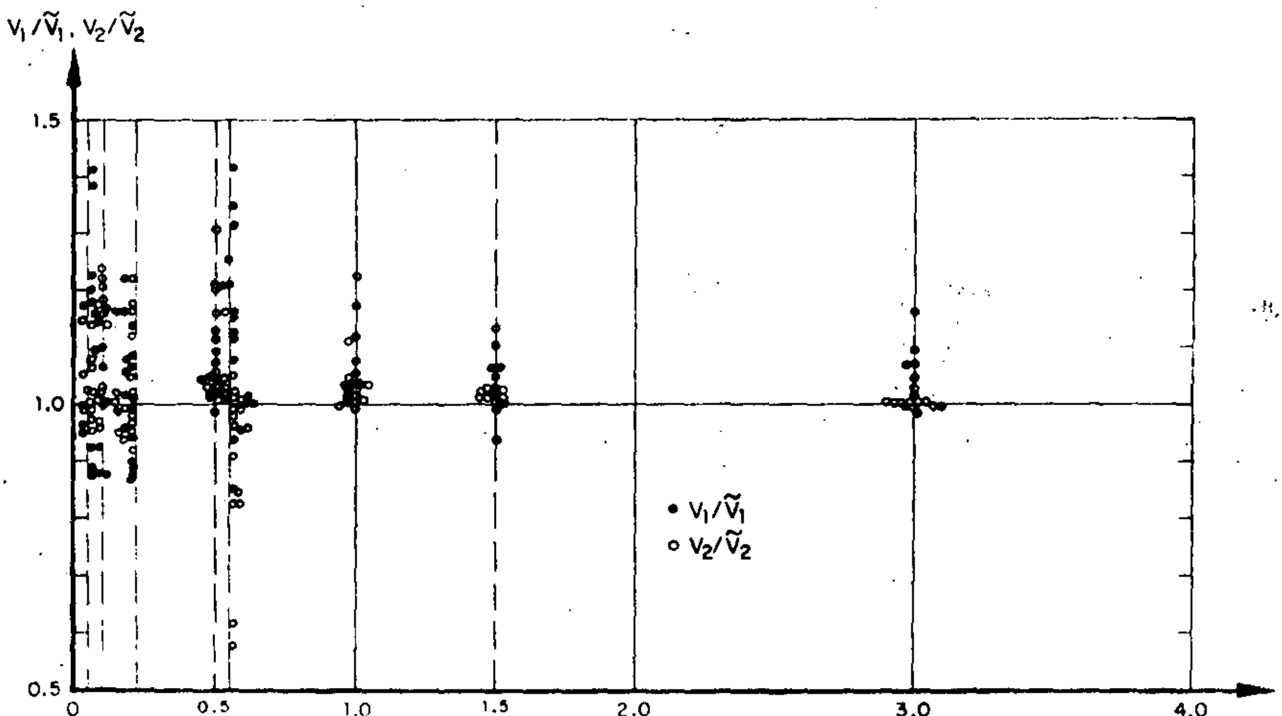


Fig 23. Fuerzas cortantes estimadas con el método 2, para $\zeta = 0.10$. Problema de traslación

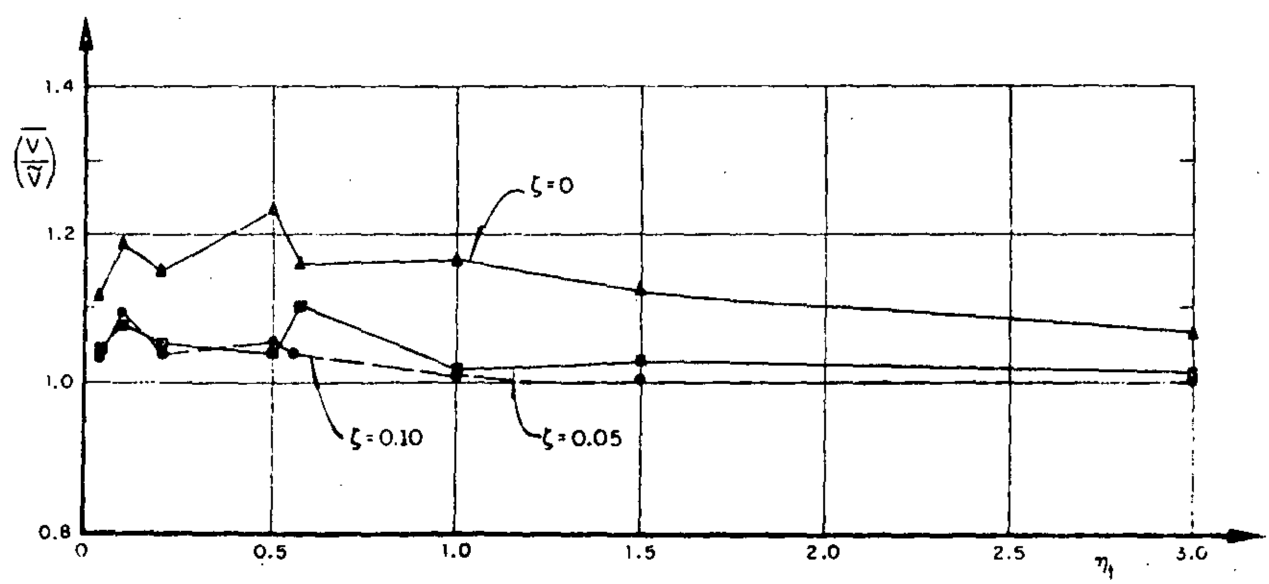


Fig 24. Variación con η_t de los promedios de las fuerzas cortantes estimadas con el método 2. Problema de traslación

3. CONCLUSIONES

El resumen de las conclusiones obtenidas de los tres problemas estudiados es:

En cabeceo y traslación:

– En promedio las estimaciones normalizadas de las respuestas máximas logradas con los métodos 1 y 2 son satisfactorias y prácticamente iguales; esto último debido a que $\epsilon_{12}^2 >> 0$

– La respuesta se subestimó con mayor frecuencia que lo que se sobrestimó, reduciéndose el error al considerar amortiguamiento en la estructura. Además, los valores exactos divididos entre los estimados fueron estadísticamente independientes de T_1 y η_c o η_t, así como del tipo de respuesta que se trate (momento de cabeceo o fuerza cortante)

En torsión:

– Las conclusiones sí difieren al tomar en cuenta el momento torsionante o la fuerza cortante. Además, debido a que en algunos casos ϵ_{12}^2 es pequeña, los dos métodos dan resultados diferentes

– Las estimaciones del momento torsionante al considerar amortiguamiento estructural nulo dependen en gran medida de la relación de frecuencias, η. Además, estos difieren al usar el método 1 o el 2, siendo más aproximados los del 1 para valores de η comprendidos en el intervalo o $0.5 \leqslant \eta \leqslant 1.5$ o muy parecidos fuera de él

– Para los tres amortiguamientos estudiados, los resultados del método 2 son estadísticamente independientes de η, no así los del 1; son mejores los del método 2 cuando $\zeta = 0.05$ y 0.10

– Cuando se tenga $0.5 \leqslant \eta \leqslant 2$, se recomienda usar el método 2; en los demás casos es indistinto el empleo de cualquiera de los dos métodos

– La relación de excentricidad dinámica a excentricidad estática se subestima en las disposiciones del Reglamento de Construcciones del Distrito Federal, siendo esto más cuando el valor de η queda comprendido entre 0.8 y 2. En particular, para $0.9 \leqslant \eta \leqslant 1.1$ esta relación vale, en promedio, 4.6 para $\zeta = 0.05$ y 2.7 para $\zeta = 0.10$. De lo anterior se concluye que es necesario realizar estudios exhaustivos sobre este aspecto, considerando vibración torsional en estructuras de varios pisos y con comportamiento inelástico

– Las distribuciones de probabilidades del cociente del valor exacto sobre el estimado son normales con desviación estándar cercana a 0.16 y media comprendida en el intervalo 1 ± 0.12 (fig 19)

APENDICE

A.1 ANALISIS DINAMICO DE UNA ESTRUCTU SUJETA A TORSION

La fig A.1 representa un edificio de un piso, de forma arbitraria, con la línea del centro de torsión (CT) al centro de gravedad (CG) perpendicular a la dirección del sismo considerado.

En dicha figura se tiene que

m masa total del sistema

J momento polar de masa respecto al centro de gravedad

L_t rigidez torsional respecto al centro de torsión

K rigidez lineal en la dirección del movimiento

e_s excentricidad estática

b dimensión de la estructura en dirección Y

$c = e_s/b$

Considerando que la rigidez torsional respecto al centro de gravedad es

$$L = L_t + K e_s^2$$

y aplicando el principio de D'Alambert para obte las ecuaciones de equilibrio del sistema en vibraciones libres, se llega al siguiente sistema de ecuaciones diferenciales lineales de segundo orden (ref 8)

$$\begin{aligned} m\ddot{z} + K(z - e_s \Phi) &= 0 \\ J\ddot{\Phi} + L\Phi - K e_s z &= 0 \end{aligned} \qquad (A.1)$$

Sustituyendo en la ec A.1 a $\ddot{z} = -\omega^2 z$ y $\ddot{\Phi} = -\omega^2 \Phi$ (por ser vibraciones libres), donde ω es la frecuencia circular natural del sistema, y resolviendo el sistema de ecuaciones algebraicas resultante, se obtiene la ecuación característica:

$$\lambda^4 - \lambda^2(1 + \eta) + \eta - c^2/j^2 = \qquad (A.2)$$

donde $\lambda^2 = \omega^2/(k/m)$, $j^2 = J/(mb^2)$ y $\eta = (L/J)/(K/m)$. Las raíces de la ec A.2 son

$$\lambda_{1,2}^2 = \frac{\eta + 1}{2} \pm \sqrt{\frac{(\eta - 1)^2}{4} + \frac{c^2}{j^2}} \qquad ($$

mientras que los vectores de las configuraciones modales son

Debido a que las conclusiones obtenidas de esas gráficas son prácticamente las mismas, en este trabajo solo se reproduce la correspondiente a las fuerzas cortantes con $\zeta = 0.10$ (fig 21). Dichas conclusiones fueron, además de las mencionadas, las siguientes:

— Los resultados son estadísticamente independientes de η con 95 por ciento de nivel de confianza, cuando $\zeta \geqslant 0.05$

— La respuesta normalizada se subestima con mayor frecuencia que lo que se sobrestima, en proporción de 2 a 1

— El error máximo en defecto fue 29 por ciento, y en exceso, 22 por ciento

— El promedio global de los resultados con $\zeta \geqslant .05$ es 1.05, y el coeficiente de variación, 10 por ciento

Los resultados varían ligeramente al introducir amortiguamiento a la estructura, se hace notar que para $\zeta = 0$, la respuesta normalizada promedio se subestima aproximadamente en 10 por ciento más que con $\zeta = 0.05$ y 0.10 (fig 22). En estos dos últimos casos no se aprecia diferencia significativa en los promedios de las respuestas ni en las dispersiones. Así, los errores máximos que se tuvieron para $\zeta = 0.05$ alcanzaron 31 por ciento en defecto y 19 por ciento en exceso; en cuanto a $\zeta = 0.10$ fueron, respectivamente, 27 y 21 por ciento

— Dado que existe gran incertidumbre en otros factores del diseño sísmico, tales como magnitud del sismo de diseño (o en las amplitudes del espectro de diseño), contenido de frecuencias, duración y variación temporal del mismo, se puede concluir que las estimaciones obtenidas con los dos métodos son, en promedio, satisfactorias en este tipo de estructuras.

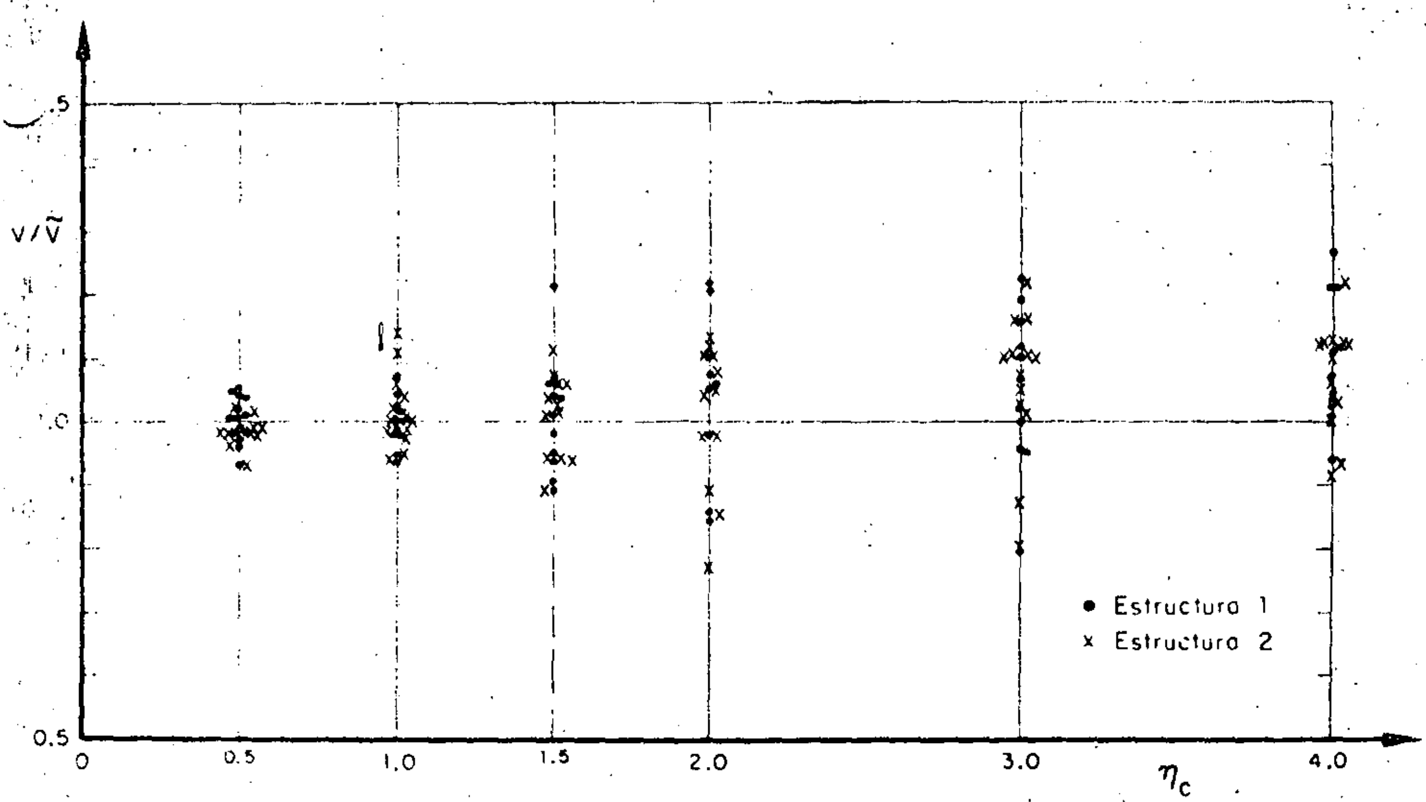


Fig 21. Fuerzas cortantes normalizadas estimadas con el método 2, para $\zeta = 0.10$. Problema de cabeceo

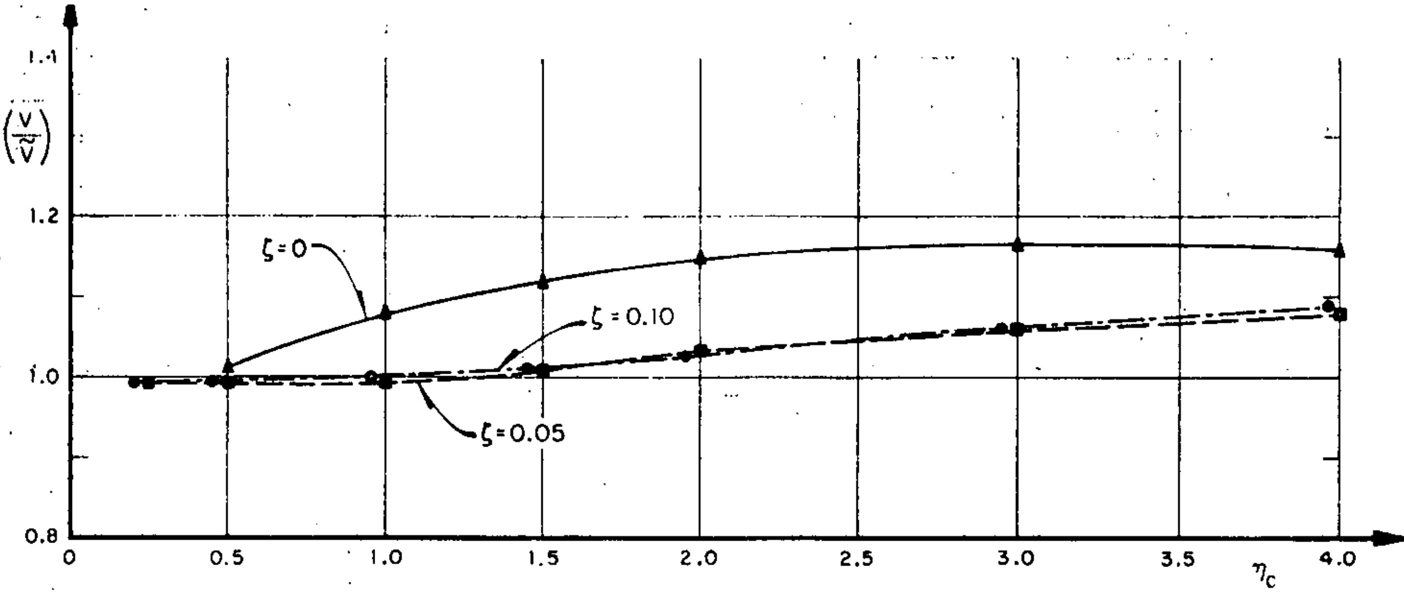


Fig 22. Variación con η_c de los promedios de las fuerzas cortantes estimadas con el método 2. Problema de cabeceo

2.3 Resultados del problema de traslación (caso 3)

Para estudiar este problema se escogieron como parámetros:

$\eta_t =$ $(k_2/m_2)/(k_1/m_1)$
T_1 periodo fundamental
ζ fracción de amortiguamiento respecto al crítico en ambos modos de vibración
m_2/m_1 relación de masas

Los valores que se asignaron a η_t fueron 0.1, 0.2, 0.5, 1.0, 1.5 y 3; a T_1, 0.3, 1.0 y 4.0 seg; a ζ, 0, 0.05 y 0.10; y a m_2/m_1, 0.5 1.0 y 2.0

Los resultados se analizaron mediante gráficas con η_t o T_1 en el eje de las abscisas, y cocientes de las fuerzas cortantes exactas entre las estimadas en el eje de las ordenadas (fuerzas cortantes normalizadas). Debido a que los resultados no difirieron mucho de los de cabeceo, se empleó únicamente un sismo como excitación. Las conclusiones a que se llegó son:

— Las estimaciones que se obtienen con los métodos 1 y 2 son prácticamente iguales, debido a que los valores de las frecuencias de vibración no resultan muy cercanas entre sí en cada caso, lo cual hace que las $\epsilon_{1,2}^2$ (ec 1.3) resulten grandes y, por tanto, que el término de las ecs A.36 y A.37 que las incluye sea muy pequeño, en cuyo caso las ecs A.34 y A.35 son casi iguales a las ecs A.36 y A.37, respectivamente. Esto se observó aun cuando se estudiaron casos adicionales de m_2/m_1 y η_t, para los cuales el radical de la ec A.29 fue mínimo, con lo cual hubo las diferencias mínimas posibles entre las dos frecuencias fundamentales y, por tanto, los valores más pequeños de $\epsilon_{1,2}^2$. Esto ocurre cuando

$$\eta_t = \frac{1 - m_2/m_1}{(1 + m_2/m_1)^2}, \text{ si } m_2/m_1 < 1$$

Dichos casos adicionales fueron: $m_2/m_1 = 0.2$ con $\eta_t = 0.555$; $m_2/m_1 = 0.5$ con $\eta_t = 0.222$, y $m_2/m_1 = 0.8$ con $\eta_t = 0.062$. En estos, la diferencia máxima que se obtuvo entre los resultados de los dos métodos fue de 13 por ciento, siendo mejores los del método 2

— Las estimaciones normalizadas son estadísticamente independientes del periodo fundamental, T_1, con nivel de confianza de 95 por ciento

— En la fig 23 se observa que las estimaciones de V_1 y V_2 tienen, en promedio, errores muy parecidos, por lo que en las conclusiones no es necesario hacer distinciones entre ellas

— La respuesta se sobrestima solamente en 30 por ciento de los casos. El error máximo en exceso que se observó fue 46 por ciento, y en defecto 41 por ciento. El coeficiente de variación para $\zeta = 0.10$ alcanzó 12 por ciento

— En la fig 24 se observa que los promedios de estimaciones con $\zeta = 0.05$ y 0.10 son mejores que que corresponden a $\zeta = 0$, lo cual hace pensar que las conclusiones obtenidas en la ref 5 respecto a $\zeta = 0$ no pueden generalizarse para $\zeta > 0$

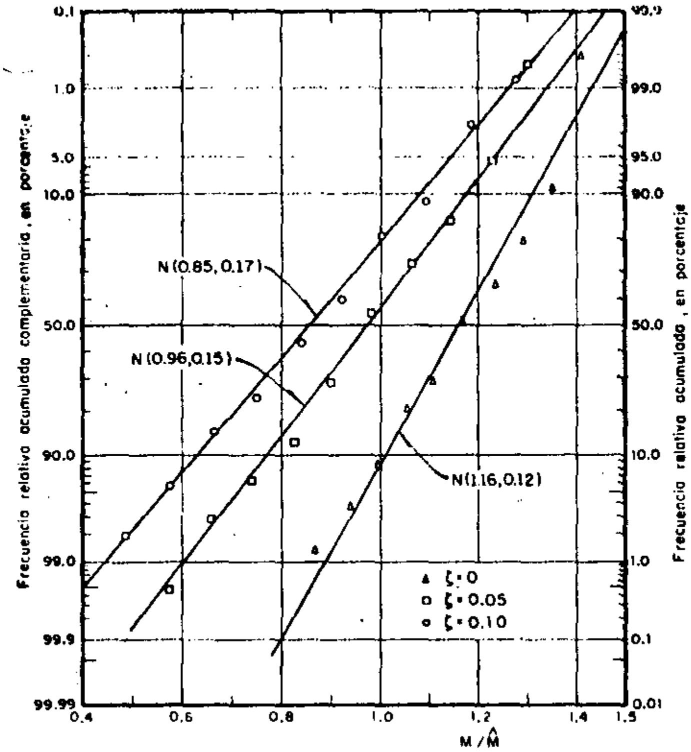


Fig 17. Resultados de los momentos torsionantes para $\eta = 1.5, 2.0, 2.5, 3.0$ y 4.0 mezclados. Papel de probabilidades normal, método 1

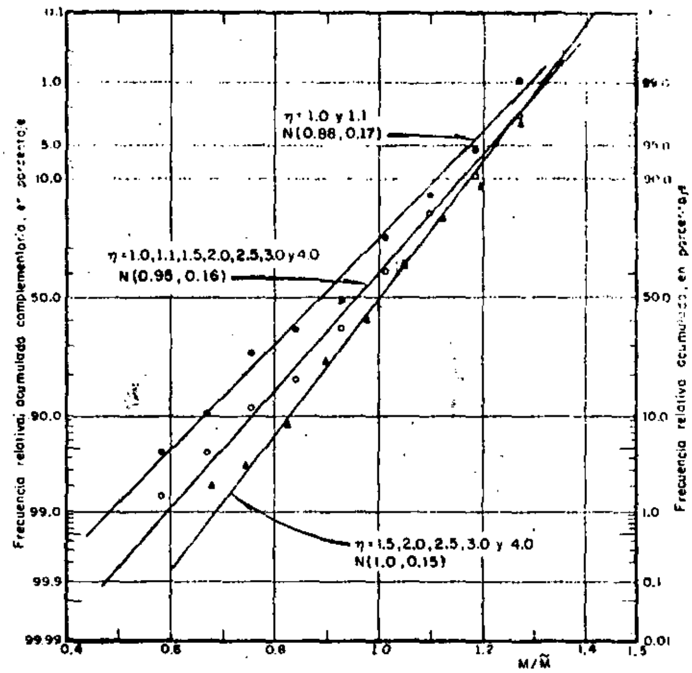


Fig 19. Resultados del método 2 para $\zeta = 0.10$

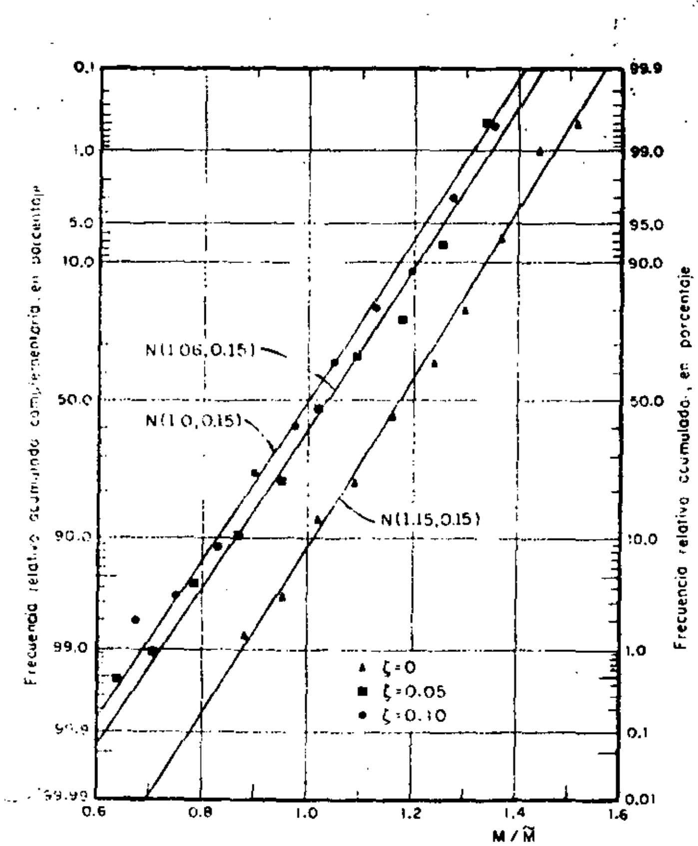


Fig 18. Resultados de los momentos torsionantes para $\eta = 1.5, 2.0, 2.5, 3.0$ y 4.0 mezclados. Papel de probabilidades normal, método 2

Para el método 1, con $\eta = 1.0$, los errores máximos fueron: 41 por ciento en defecto para la estructuración del caso I y 32 por ciento en defecto en los casos II y III. Los errores medios respectivos fueron 36 y 15 por ciento, ambos en defecto. Para $\eta = 1.1$, la estructuración del caso I tuvo errores máximos de ± 5 por ciento, y las tipo II y III, 38 por ciento en defecto y 11 por ciento en exceso.

Respecto al amortiguamiento, se concluyó que las fuerzas cortantes normalizadas son prácticamente independientes de este; así, para $\eta = 1$, los promedios globales de los métodos 1 y 2 fueron 1.23 y 1.11, respectivamente, para $\zeta = 0$; para $\zeta = 0.05$ de 1.30 y 1.02, y para $\zeta = 0.10$ de 1.30 y 1.0.

Como puede apreciarse mediante los promedios citados en el párrafo anterior, los resultados que se obtienen con el método 2 son mejores que los del 1 cuando $\eta = 1.0$. Una conclusión semejante se obtuvo cuando $\eta = 0.9$ y 1.1, aunque las diferencias se redujeron en un 10 por ciento. Para valores de η fuera del intervalo $0.9 \leq \eta \leq 1.1$, los resultados de ambos métodos fueron prácticamente iguales.

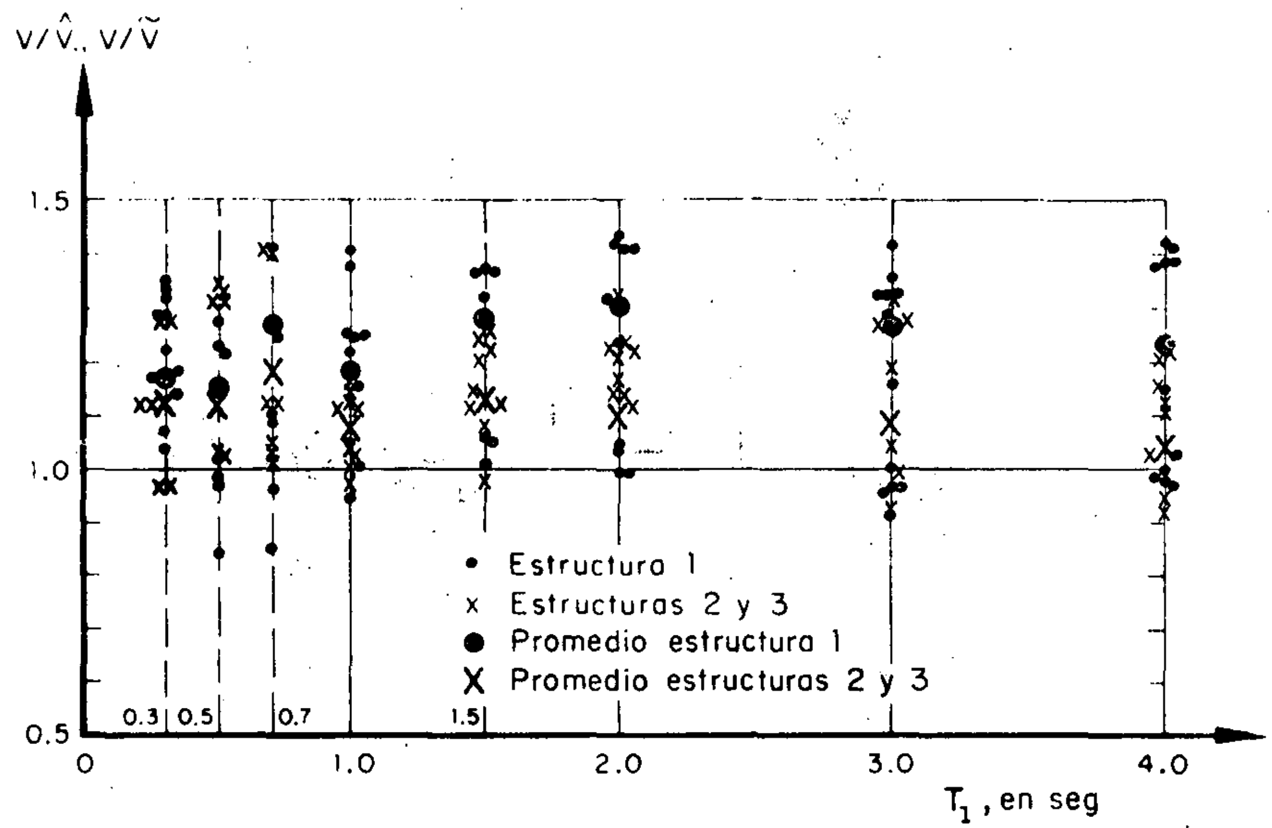


Fig 20. Fuerzas cortantes normalizadas estimadas con los métodos 1 y 2, para $\eta = 1.0$ y $\zeta = 0$

2.2 Resultados del problema de cabeceo (caso 2)

Los parámetros que se escogieron para estudiar el problema de cabeceo fueron:

- m masa total
- L distancia del suelo al centro de gravedad
- T_1 periodo fundamental periodo
- ζ fracción de amortiguamiento respecto al crítico en ambos modos de vibración
- η_c cociente de la frecuencia angular entre la lineal

Los valores que se asignaron a T_1 fueron 0.3, 0.7, 1.0, 1.5, 2.0, 3.0 y 4.0 seg; a ζ, 0, 0.05 y 0.10, y a η_c, 0.2, 0.5, 1.0, 1.5, 2.0, 2.5, 3.0 y 4.0. En cuanto a m y L, únicamente se usaron 2.0 ton seg²/m y 4 m, respectivamente, ya que por los resultados (fuerzas y momentos normalizados) que se obtuvieron con estas combinaciones se juzgó innecesario el uso de otros valores; por la misma razón se emplearon únicamente tres de los sismos del problema de torsión.

En este problema, igual que en el de torsión, no hubo diferencias apreciables entre los resultados obtenidos con los tres sismos que se emplearon como excitación, por lo cual se agruparon los resultados en una sola muestra. Además, tanto las fuerzas cortantes como los momentos de cabeceo máximos normalizados fueron estadísticamente independientes del periodo fundamental, T_1, con nivel de confianza de 95 por ciento.

Otra conclusión interesante es que los resultados obtenidos con los métodos 1 y 2 (Apéndice) son prácticamente iguales, con diferencias máximas entre ellos de 5 por ciento. Esto se debe a que los valores de ϵ_{12}^2 (ec 1.3) son grandes porque las frecuencias de vibración no resultan con valores muy cercanos entre sí, aun cuando se usaron η_c muy pequeñas, de manera que el radical de la ec A.17 fuera también pequeño y, por tanto, que las diferencias entre las dos frecuencias fundamentales fueran mínimas. Esto hace que los términos que contienen a ϵ_{12}^2 en las ecs A.24 y A.25 resulten muy pequeños y que estas ecuaciones sean casi iguales a las ecs A.22 y A.23, respectivamente.

Aprovechando las conclusiones anteriores, se acumularon las muestras correspondientes a todos los periodos fundamentales, y para cada amortiguamiento se elaboraron dos gráficas: una de fuerzas cortantes y otra de momentos de cabeceo normalizados, empleando únicamente los resultados del método 2. En f[...] el eje de las abscisas representó a η_c, y el de las c[...]nadas a los cocientes $V/\tilde{V}$ o $M/\tilde{M}$, donde V y M o[...] tan la fuerza cortante y el momento de cabeceo exactos, y $\tilde{V}$ y $\tilde{M}$ los mismos elementos mecánicos estimados con el método 2.

Se observa en la fig 14, que corresponde a amortiguamiento nulo, que para $\eta = 0.9$, 1.0 y 1.1 hay una marcada diferencia entre los resultados obtenidos para el caso I con los casos II y III (la de estos últimos entre sí no es tan importante). Así, cuando $\eta = 1.0$, en el caso I el promedio de e_d/e_s fue 38.5 y la desviación estándar 16.6; en el caso II estos parámetros estadísticos valieron 5.4 y 0.6, respectivamente. Para valores de η separados de 1.0 en 0.5 unidades o más hay diferencias menos apreciables entre los resultados de los tres casos. Además, e_d/e_s disminuye rápidamente conforme η se aleja de 1.0.

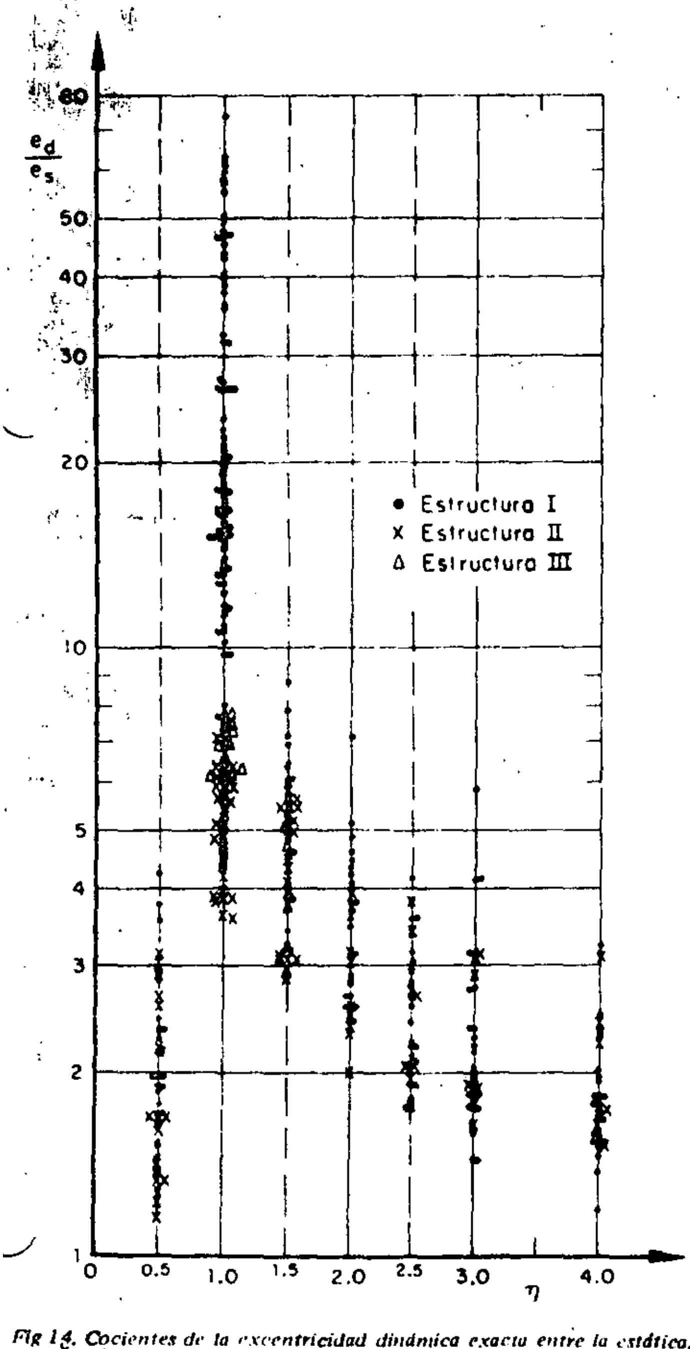


Fig 14. Cocientes de la excentricidad dinámica exacta entre la estática, para $\zeta = 0$

En las figs 15 y 16, para $\zeta = 0.05$ y 0.10, respectivamente, casi no hay diferencias entre los resultados de los dos casos, aunque persiste la dependencia respecto a η. Comparando estas tres últimas figuras se nota también que e_d/e_s disminuye conforme el amortiguamiento aumenta. Así, para $\zeta = 0.05$ el promedio fue 4.6 y la desviación estándar 1.3, mientras que para $\zeta =$ = 0.10, los valores correspondientes fueron 2.7 y 0.7

De las figs 15 y 16 se concluye que la disposición del Reglamento de Construcciones del Departamento del Distrito Federal de que se tome $e_d/e_s = 1.5$ subestima el valor promedio para todos los valores de η mayores de 0.5 y menores de 4.0 (aquí se omitió el término ± 0.05b que se agrega a 1.5 en la disposición del Reglamento, porque dicho término tiene como finalidad prevenir excentricidades accidentales ocasionadas por variaciones imprevisibles de masas y rigideces y posibles excitaciones torsionales).

Con objeto de estimar probabilidades de eventos relacionados con los momentos torsionantes, se trazaron en papel de probabilidades los datos de frecuencias acumuladas correspondientes a diferentes casos. Las distribuciones de probabilidades empleadas fueron la logarítmico normal, la extrema tipo II y la normal, de las cuales, por apreciación visual, se consideró que esta última daba en general mejores resultados (figs 17 a 19).

Para verificar que las poblaciones bajo estudio tienen distribuciones normales, se realizaron pruebas de hipótesis estadísticas con un 95 por ciento de nivel de confianza.

Los resultados fueron:

Método 1

(Con resultados de $\eta = 1.5$, 2.0, 2.5, 3.0 y 4.0 mezclados; fig 17)

$\zeta = 0$: se rechaza la hipótesis nula de que la distribución es normal con media 1.16 y desviación estándar 0.12 (esta hipótesis se rechaza también con un 99 por ciento de nivel de confianza)

$\zeta = 0.05$ y $\zeta = 0.10$: se aceptan las hipótesis nulas de que las distribuciones son normales con medias 0.96 y 0.85, y desviaciones estándar 0.15 y 0.17, respectivamente.

Método 2

(Con resultados de $\eta = 1.5$, 2.0, 2.5, 3.0 y 4.0 mezclados; fig 18)

$\zeta = 0$, 0.05 y 0.10: se aceptan las hipótesis de que las distribuciones son normales con medias 1.15, 1.06 y 1.00, y desviaciones estándar 0.15, 0.15 y 0.15, respectivamente. Para $\zeta = 0.05$, la hipótesis se acepta con 99 por ciento de nivel de confianza; las otras con 95 por ciento.

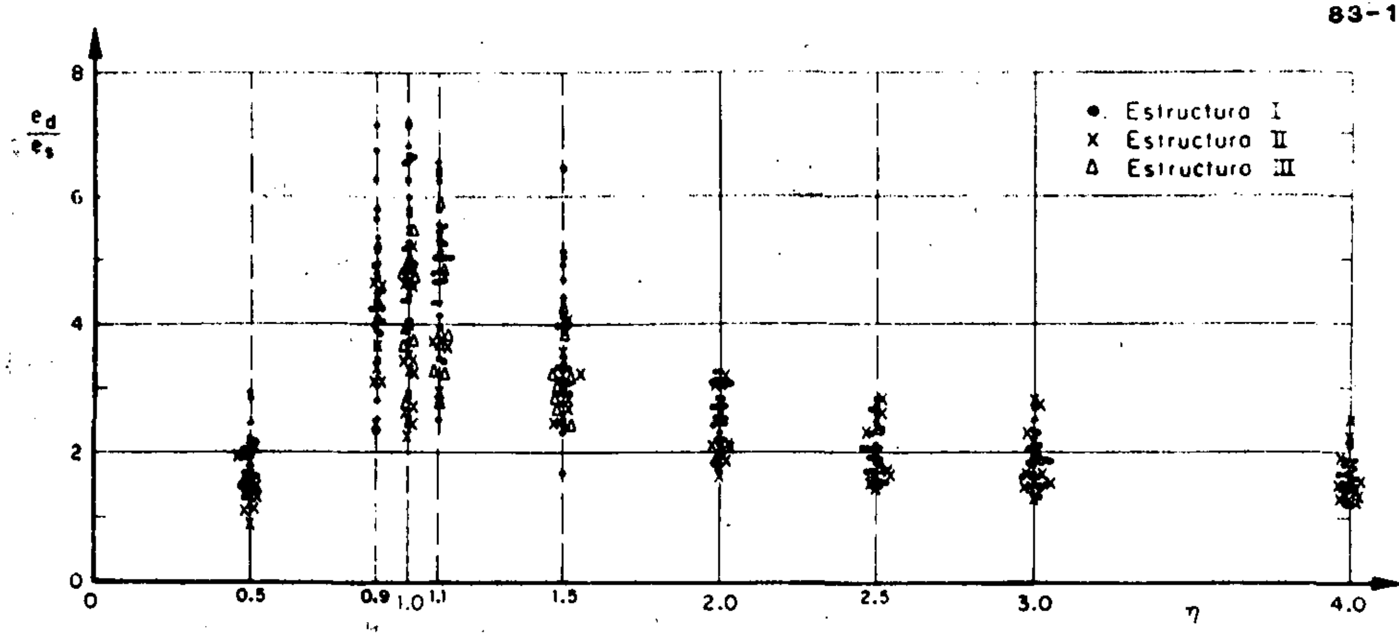


Fig 15. Cocientes de la excentricidad dinámica exacta entre la estática, para $\zeta = 0.05$

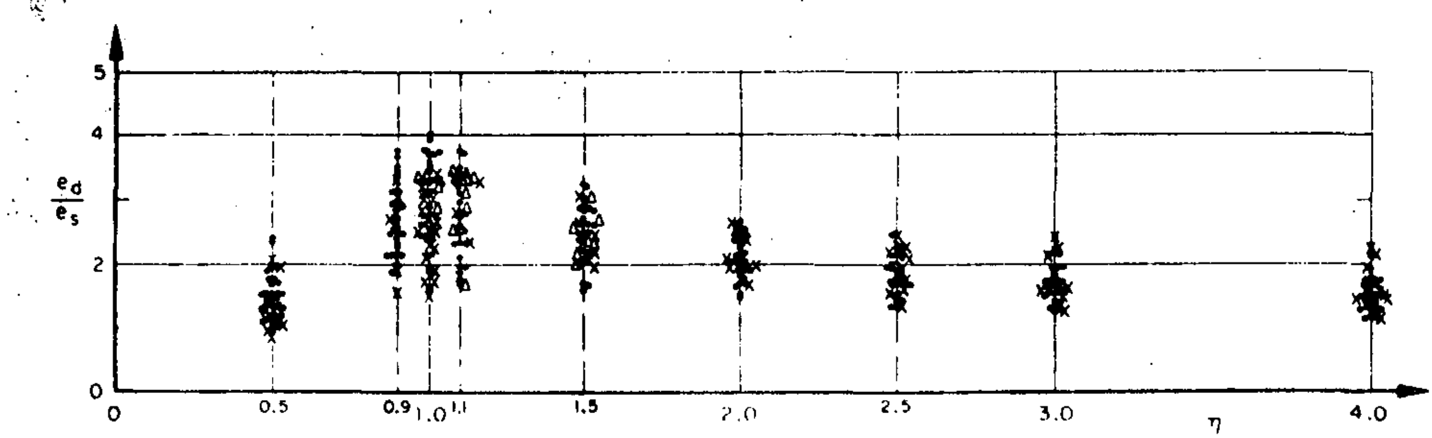


Fig 16. Cocientes de la excentricidad dinámica exacta entre la estática, para $\zeta = 0.10$

Además, para $\zeta = 0.10$ se estudió el caso en que se mezclaron los resultados de $\eta = 1$ y $\eta = 1.1$ (fig 19), obteniéndose una distribución normal con media 0.88 y desviación estándar 0.17. También se mezclaron los resultados de los valores de η de 1 a 4, para los cuales se obtuvo una distribución de igual tipo con media 0.95 y desviación estándar 0.16. Ambas hipótesis fueron aceptables, pero con 97.5 por ciento de nivel de confianza.

En todos los casos descritos en que se acepta la hipótesis nula, se observa que la desviación estándar es muy semejante, ya que varía de 0.15 a 0.17, mientras que la media va de 0.86 a 1.15.

2.1.2 Fuerza cortante

Los resultados obtenidos con los métodos 1 y 2, correspondientes a $\eta = 1.0$ y $\zeta = 0$, se muestran en la fig 20. En el eje de las abscisas se tienen los periodos fundamentales, T_1, y en el de las ordenadas las fuerzas cortantes normalizadas, $V/\hat{V}$ y $V/\tilde{V}$, obtenidas al dividir las fuerzas cortantes, V, calculadas mediante análisis modal entre las estimadas con los métodos 1 y 2, $\hat{V}$ y $\tilde{V}$, respectivamente.

De la fig 20 y otras similares se concluyó que las fuerzas cortantes normalizadas obtenidas con ambos métodos son independientes del periodo fundamental, T_1, con 95 por ciento de nivel de confianza. Además, para valores de η menores de 0.9 y mayores de 1.1, los resultados fueron independientes de los parámetros A, b y c, con errores de ± 5 por ciento. Esta independencia también se obtuvo para el método 2, inclusive cuando $\eta = 0.9$, 1.0 y 1.1, con errores máximos de 40 por ciento en defecto y 20 por ciento en exceso para $\zeta = 0$, tendiendo a reducirse conforme aumenta el amortiguamiento; así, para $\zeta = 0.05$, se obtuvieron errores máximos de ± 20 por ciento, y para $\zeta = 0.10$ de ± 10 por ciento.

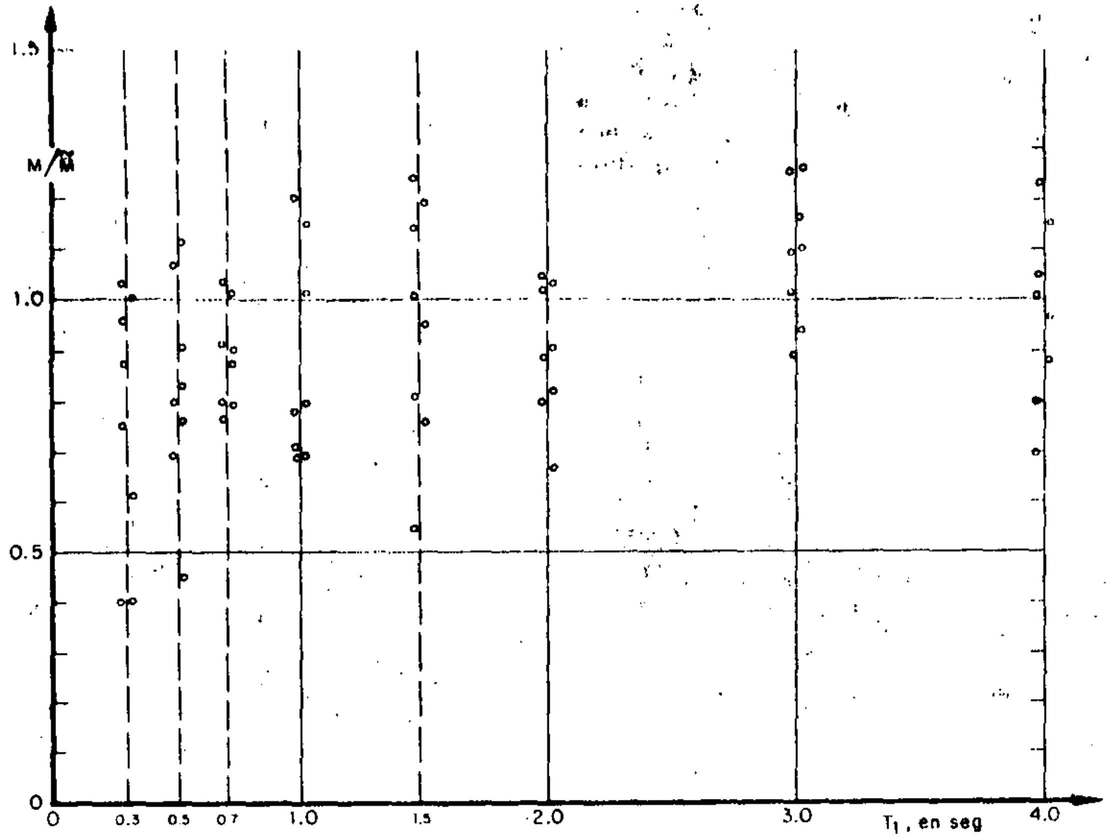


Fig 11. Resultados de los momentos torsionantes para $\eta = 1.0$, y $\zeta = 0.05$. Método 2

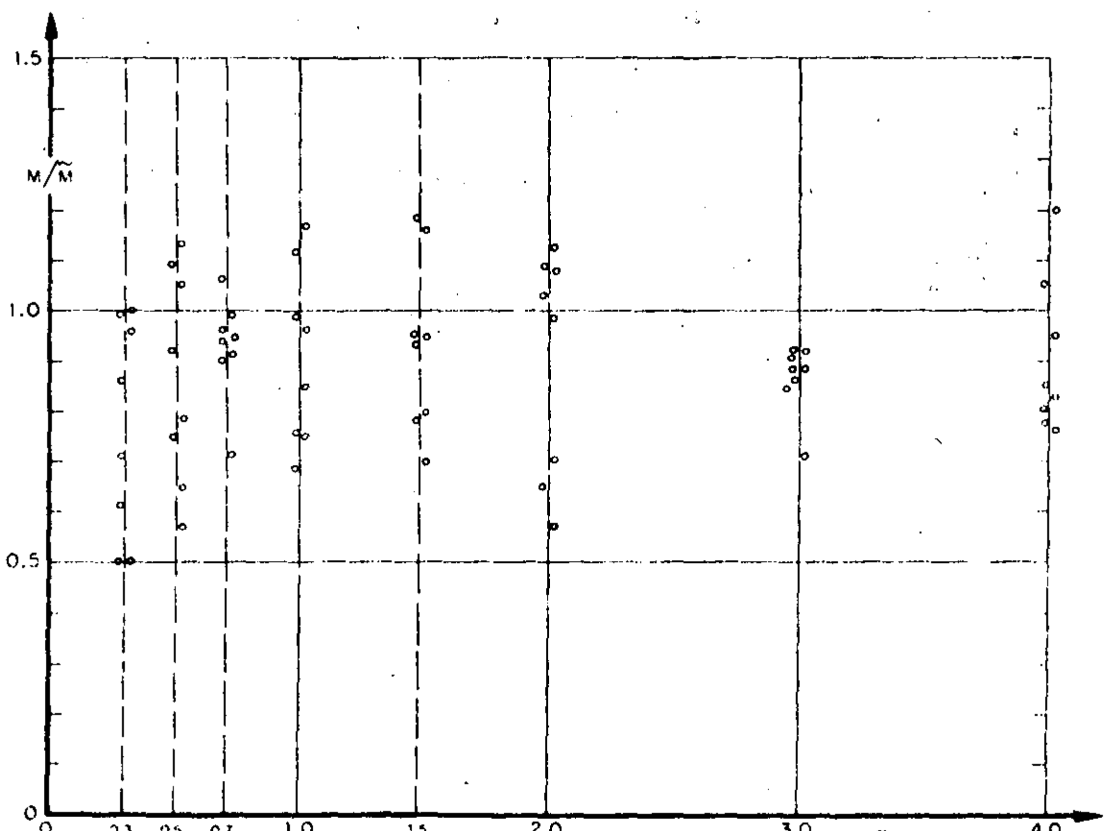


Fig 12. Resultados de los momentos torsionantes para $\eta = 1.0$, y $\zeta = 0.10$. Método 2

En la fig 13 se presentan en el eje de las ordenadas los promedios, $(M/\hat{M})$ y $(M/\tilde{M})$, de los resultados obtenidos respectivamente con los métodos 1 y 2, considerando que estos son independientes de T_1; en el eje de las abscisas se localizan los valores de η. Se observa que, para η = 0.9, 1.0 y 1.1, el método 2 sobrestima ligeramente la respuesta media (en 10 por ciento), tendiendo a subestimarla en 5 por ciento conforme los valores de η se alejan de 1.0, cuando ζ = 0.05 y 0.10

Con objeto de verificar si con el método 2 los resultados son independientes de η, se realizaron pruebas de hipótesis de igualdad de medias, siendo aceptables con 95 por ciento de nivel de confianza. Por lo contrario, los resultados del método 1 no fueron independientes de η, lo cual es obvio, puesto que con ζ = = 0.10 se tiene que el promedio de $M/\hat{M}$ es 0.31 para η = 1 (el mínimo valor fue 0.04 y el máximo 0.68), y 0.99 para η = 4 (el mínimo fue 0.66 y el máximo 1.28).

En la fig 13 se observa también que los promedios obtenidos con el método 1 se acercan a los exactos conforme η aumenta, presentándose mayores errores para valores de η muy cercanos a 1.0, para el cual las frecuencias naturales de la estructura resultan más próximas entre sí (ec A.3), lo que trae como consecuencia que en muchas ocasiones las respuestas máximas en ambos modos de vibración ocurran simultáneamente y con signo contrario, por lo que la respuesta combinada máxima es la suma algebraica de ambas respuestas, que da resultados menores que los de la ec A.11.

Otra conclusión inmediata que se obtiene de la fig 13 es que los resultados del método 2 son prácticamente independientes de ζ cuando $\zeta \geq$ 0.05 y que el método 1 pierde aproximación conforme aumenta ζ, y η se aproxima a 1

De lo anterior se concluye también que en estructuras amortiguadas, que son las de interés práctico, el método 2 proporciona, en promedio, mejores resultados que el método 1, aunque el 2 subestime más y con mayor frecuencia la respuesta máxima. En estructuras no amortiguadas, que únicamente son de interés académico, el método 1 proporciona mejores resultados.

Otro punto importante de discusión es el del cociente de la excentricidad dinámica exacta, e_d, entre la estática, e_s. En las figs 14 a 16 se tiene η en el eje de las abscisas, y e_d/e_s en el eje de las ordenadas.

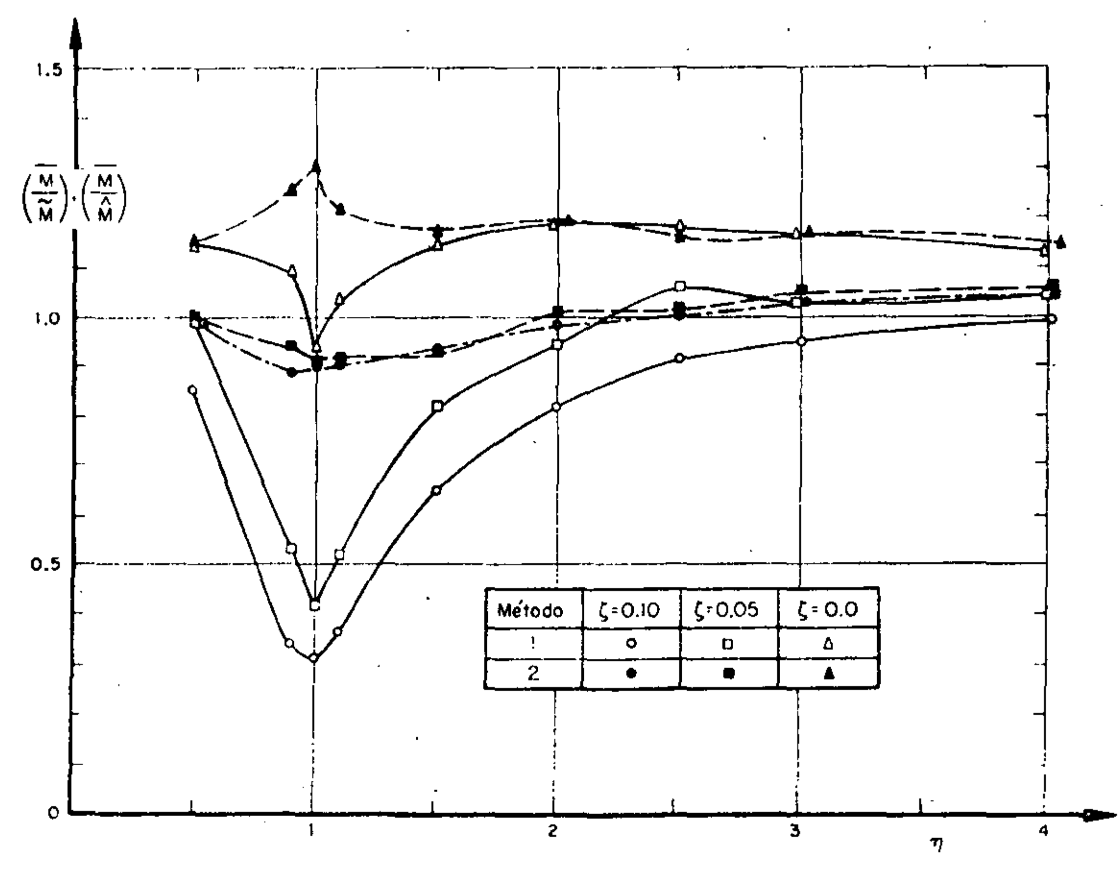


Fig 13. Variación con η de los promedios de los momentos torsionantes estimados

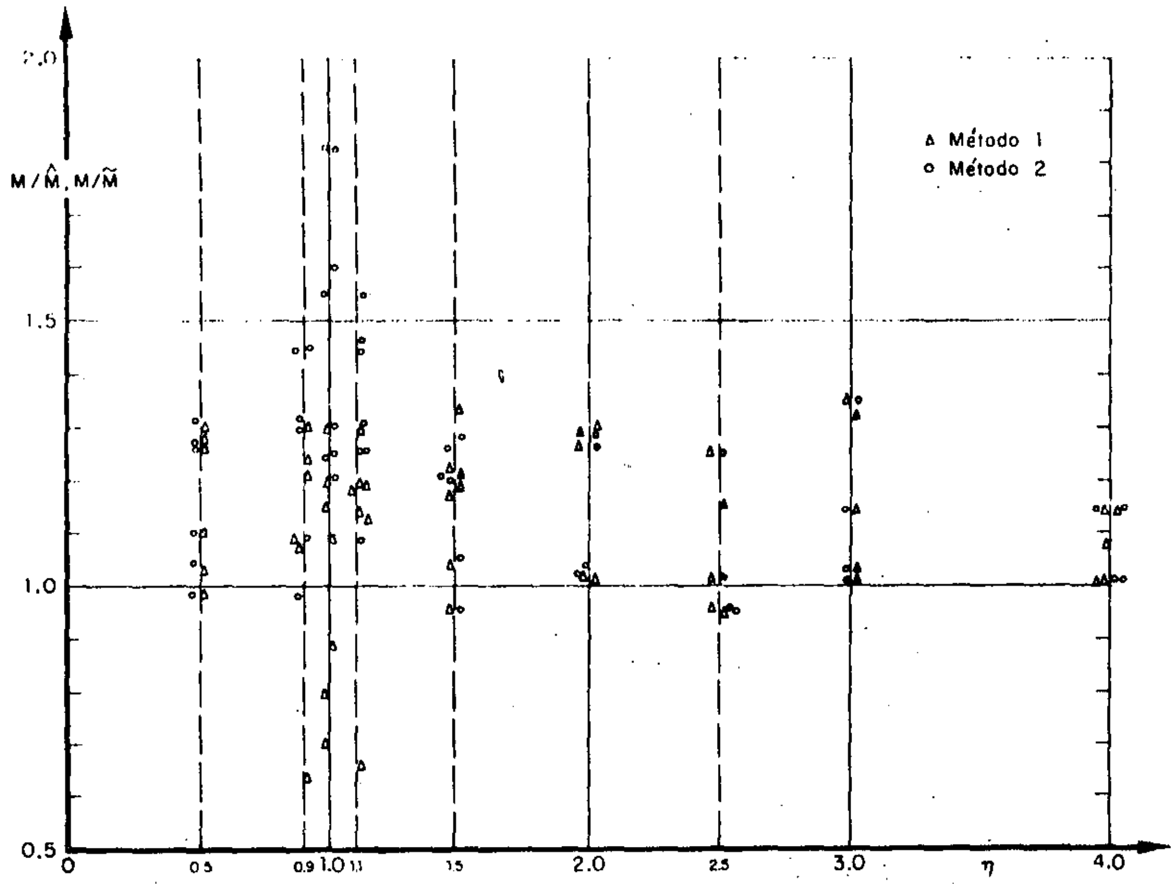


Fig 7. Resultados de los momentos torsionantes para $T_1 = 2.0$ seg, y $\zeta = 0$

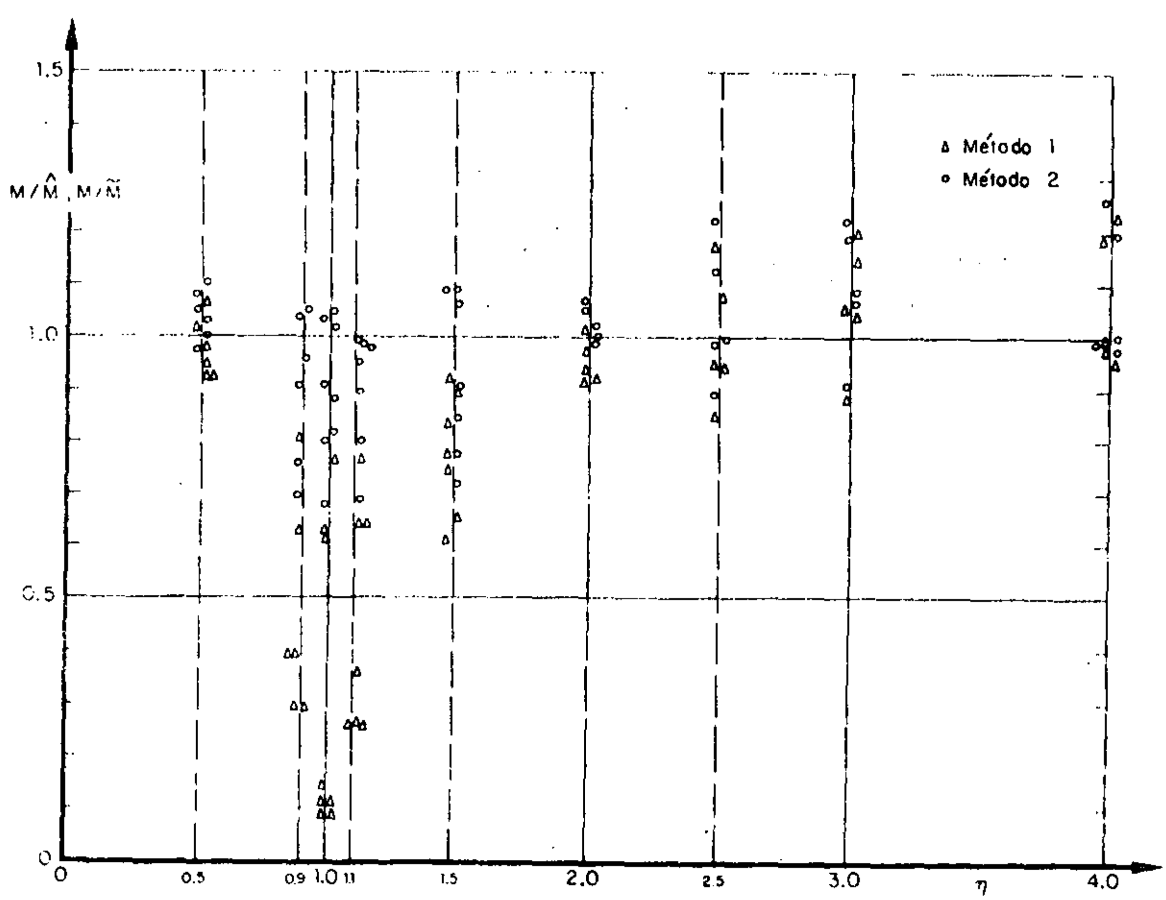


Fig 8. Resultados de los momentos torsionantes para $T_1 = 2.0$ seg, y $\zeta = 0.05$

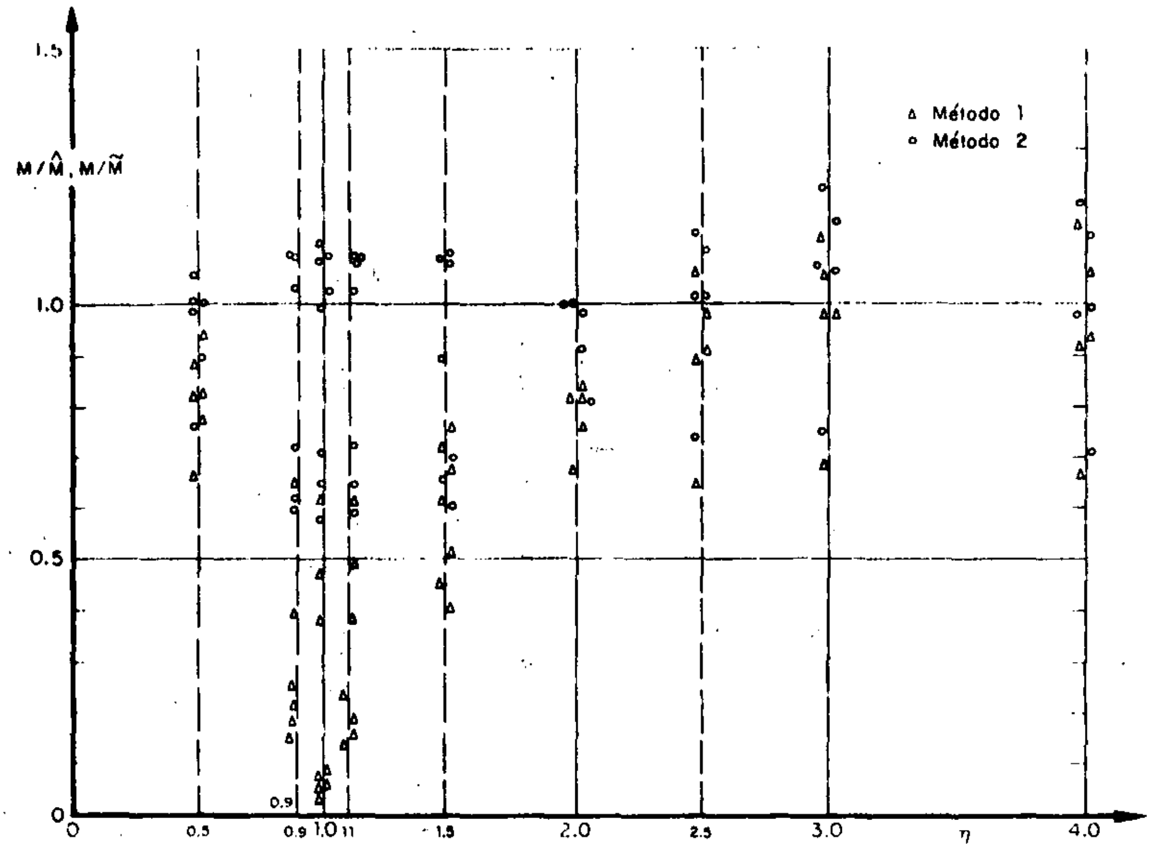


Fig 9. Resultados de los momentos torsionantes para $T_1 = 2.0$ seg, y $\zeta = 0.10$

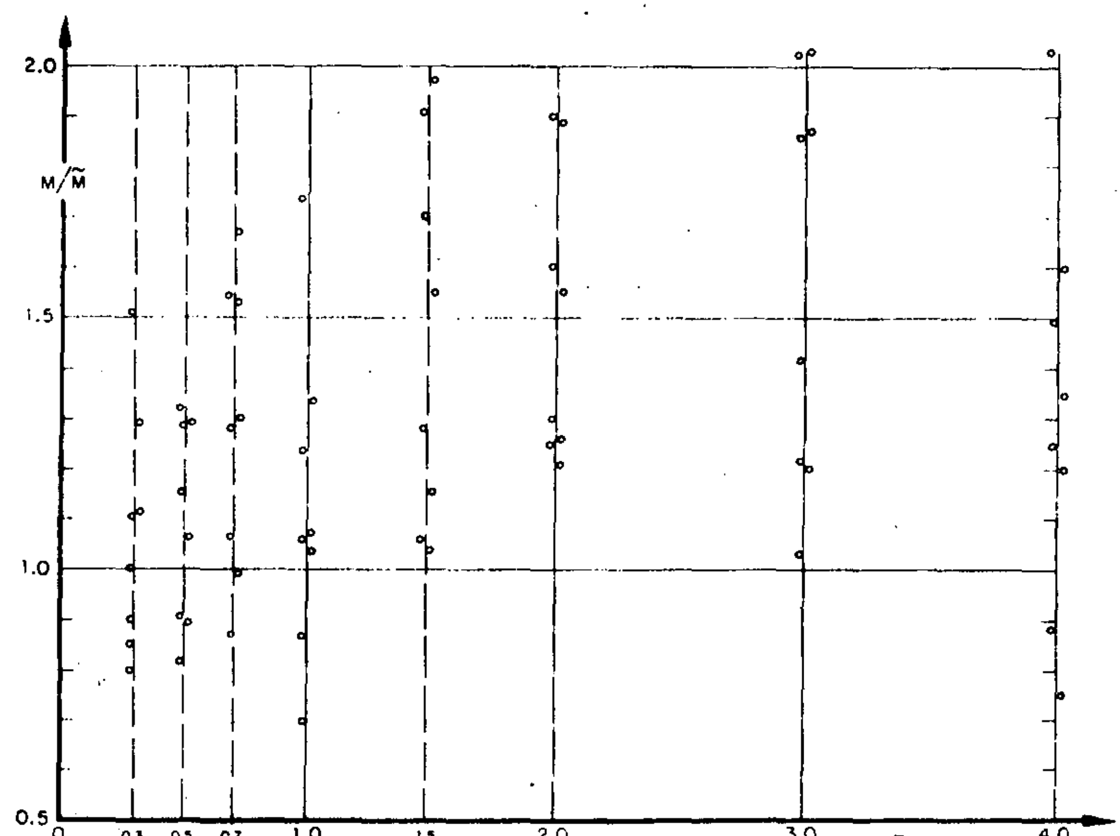


Fig 10. Resultados de los momentos torsionantes para $\eta = 1.0$, y $\zeta = 0$. Método 2

96

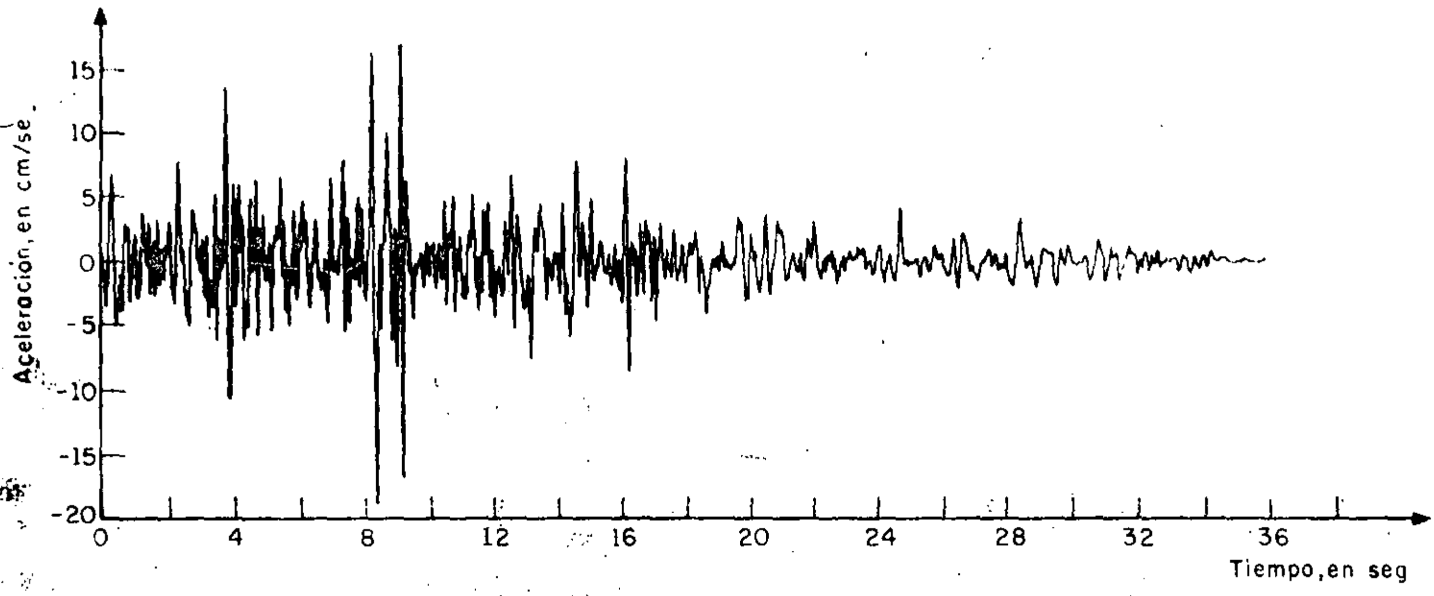


Fig 4. Sismo simulado No. 4

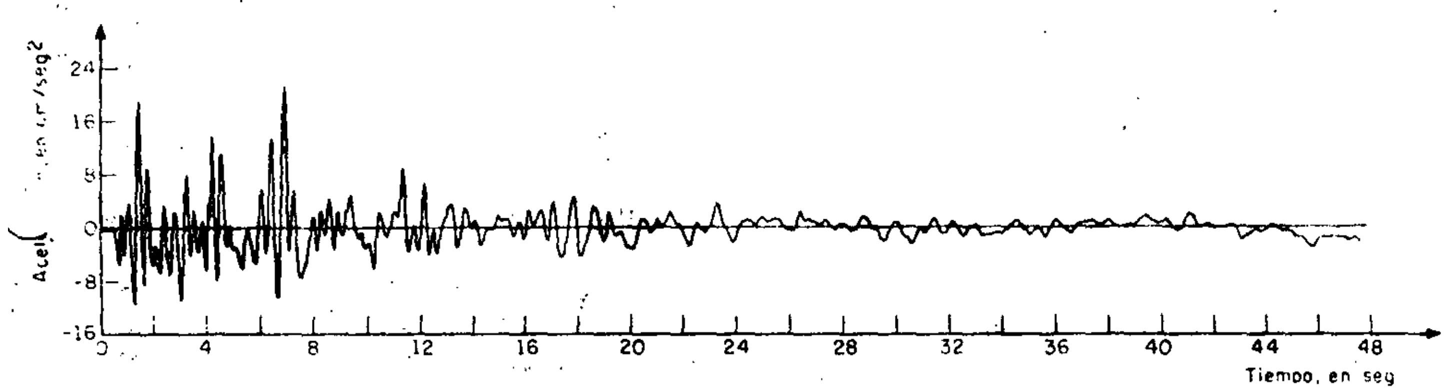


Fig 5. Sismo real registrado en la Alameda Central, México, D. F., el 10 de diciembre de 1961

a) Asignar diversos valores a cada parámetro que interviene en el problema, de manera que se cubran los intervalos de interés de cada uno.

b) Calcular la respuesta máxima exacta y las estimadas con los métodos 1 y 2 para cada combinación de valores de los diferentes parámetros.

c) Obtener las respuestas normalizadas dividiendo los valores exactos entre los estimados; esto se hace para cada combinación de valores de los parámetros, con lo cual se elimina la dispersión en los resultados ocasionada por la magnitud y variación con el tiempo de los datos de entrada (se reduce la variancia).

d) Estudiar si existen diferencias estadísticas significativas entre los resultados obtenidos al variar los valores asignados a uno de los parámetros. Si las hay, se infiere que los resultados logrados con cada valor de dicho parámetro corresponden a poblaciones estadísticas diferentes; en caso contrario, la población estadística es la misma y, por consiguiente, las muestras respectivas pueden agruparse en una sola de mayor tamaño, a partir de la cual es factible obtener conclusiones más generales y confiables acerca del modelo en estudio, ya que la variancia del promedio de la estimación se reduce en proporción a $1/n$ (ref 11). Esta etapa se repite sucesivamente para cada uno de los parámetros restantes, con lo que se realiza, de hecho, un análisis de variancia.

2.1 Resultados del problema de torsión (caso 1)

Para diseño sísmico de edificios, los elementos mecánicos que usualmente interesa conocer son las fuerzas y momentos que obran sobre cada elemento estructural. Para simplificar, con objeto de aislar los efectos de la fuerza cortante y del momento torsionante, en este problema de torsión se considerará una estructura (fig 6) con masa uniformemente distribuida, con un solo muro en dirección *Z* que resista la fuerza cortante directa, y dos idénticos en dirección *Y* (per-

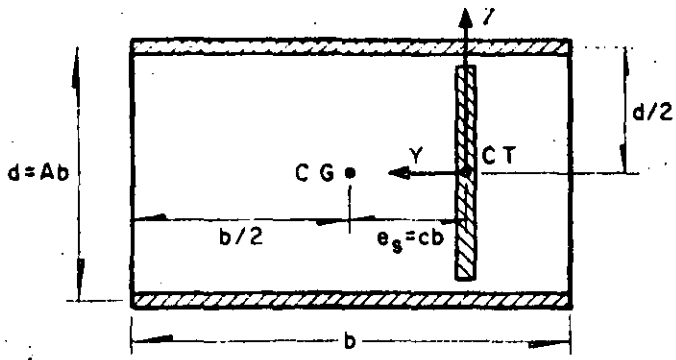


Fig 6. Estructura tipo considerada en el problema de torsión

pendicular al movimiento), de manera que cada uno de estos últimos resista una fuerza cortante igual a ***M/d***, donde *M* es el momento torsionante dinámico y *d* es la separación de los dos muros. En este caso, la estructura presenta excentricidad solo en dirección perpendicular a la de excitación, *Z*.

Los parámetros que se escogieron para estudiar el problema de torsión fueron (fig 6):

$A = b/d$

b dimensión en la dirección *Y*

$c = e_s/b$

T_1 periodo fundamental de vibración $= \omega_1/2\pi = \lambda_1/(2\pi K/m)$

ζ fracción de amortiguamiento respecto al crítico en ambos modos de vibración

η cociente de la frecuencia angular entre la lineal $= (L/J)/(K/m)$

Los valores que se asignaron a ***A, b*** y ***c*** son los consignados en la tabla 1; los de ζ son 0, 0.05 y 0.10; los de η, 0.5, 0.9, 1.0, 1.1, 1.5, 2.0, 2.5, 3 y 4, y los de T_1, 0.1, 0.3, 0.5, 0.7, 1.0, 1.5, 2, 3 y 4 seg. Los casos de η = 1, 0.9 y 1.1 se estudiaron con especial cuidado debido a que para valores de $\eta = 1$ y cercanos, sucede que las dos frecuencias naturales de vibración resultan más próximas entre sí (ec A.3) y, en consecuencia, el término ϵ_{12}^2 de las ecs A.8 y A.9 del Apéndice puede asumir valores pequeños (ec 1.3), en cuyo caso se pueden presentar diferencias considerables entre los resultados de ambos métodos, puesto que el término de la doble suma de la ec 1.2 asume valores tanto mayores cuanto menores son los de ϵ_{12}^2.

Para cada uno de los casos de la tabla 1 se obtuvieron las fuerzas cortantes y los momentos torsionantes máximos correspondientes a todas las combinaciones de ζ, T_1 y η.

En las figuras que aparecen más adelante no se hace distinción de los resultados obtenidos con cada sismo ni con cada combinación de *A, b* y *c*, ya que las muestras respectivas se mezclaron al no haberse encontrado diferencias estadísticas significativas con un 95 por ciento de nivel de confianza en los mism a pesar de la marcada diferencia entre los valores de dichos parámetros y de las características de los sismos, tales como duración y frecuencia dominante.

2.1.1 Momento torsionante

En las figs 7 a 9 se presentan los resultados correspondientes a los casos en los que T_1 = 2.0 seg y ζ = = 0, 0.05 y 0.10, respectivamente. En el eje de las abscisas se localizan los valores de η, y en el de las ordenadas los cocientes de los momentos torsionantes exactos, M, entre los estimados, $\hat{M}$ y $\tilde{M}$, con los métodos 1 y 2, respectivamente (Apéndice).

En la fig 7, en la que el amortiguamiento es nulo, se aprecia mayor dispersión en los resultados de ambos métodos que corresponden a η = 0.9, 1.0 y 1.1 que para los demás valores de η. En cambio, en las figs 8 y 9, que corresponden a ζ = 0.05 y ζ = 0.10, respectivamente, se observa que la dispersión de los resultados del método 2 es prácticamente la misma para todos los valores de η (el coeficiente de variación es cercano a 0.2), cosa que no sucede con los resultados del método 1, para los cuales se tiene mayor dispe sión cuando η = 0.9, 1.0 y 1.1. Estas observacio llevan a la conclusión de que para el método 1 n pueden mezclar las muestras correspondientes a todos los valores de η, ya que los resultados dependen de este parámetro, mientras que para el método 2 podrían mezclarse las que no se refieren a amortiguamiento nulo si se verificara que los valores medios correspondientes a cada η son estadísticamente iguales.

Para lograr dicha verificación, se investigó primero si los resultados del método 2 son independientes del periodo fundamental, T_1. Con este fin se trazó un juego de figuras del mismo tipo que las figs 10 a 12, que corresponden a η = 1.0 con ζ = 0, 0.05 y 0.10, respectivamente. En la fig 10, que corresponde a ζ = = 0, se observa que los resultados sí dependen de T_1, ya que los valores medios son sensiblemente más grandes para periodos mayores de 1.0 seg que para los menores. Por lo contrario, en las figs 11 y 12 se nota que los valores medios son prácticamente independientes de T_1 en el intervalo de periodos estudiado, por lo que las muestras de cada periodo pueden agruparse en una sola (esta conclusión también es válida para los resultados del método 1).

Para verificar estadísticamente la conclusión ante se realizó una prueba de hipótesis acerca de si la diente de la recta que se ajusta a los datos puede considerarse nula, habiéndose aceptado con 95 por ciento de nivel de confianza.

donde n es el total de grados de libertad del sistema.

El método 2 consiste en aplicar la fórmula

$$Q = \sqrt{\sum_{i=1}^{n} Q_i^2 + \sum_{i \neq j} \sum \frac{Q_i\ Q_j}{1 + \epsilon_{ij}^2}} \tag{1.2}$$

siendo

$$\epsilon_{ij} = \frac{\omega_i' - \omega_j'}{\zeta_i'\ \omega_i + \zeta_j'\ \omega_j} \tag{1.3}$$

donde

Q_i	respuesta máxima en el i-ésimo modo de vibración, tomada con el mismo signo que el de la correspondiente función de transferencia cuando esta alcanza su valor máximo absoluto
ω_i	i-ésima frecuencia circular natural de vibracion del sistema sin amortiguamiento
$\omega_i' = \omega_i \sqrt{1 - \zeta_i^2}$	i-ésima frecuencia circular natural de vibración del sistema amortiguado
ζ_i	fracción del amortiguamiento crítico en el i-ésimo modo natural
$\zeta_i' = \zeta_i + 2/(\omega_i S)$	fracción del amortiguamiento crítico equivalente
S	duración del sismo con el que se excita al sistema

El interés primordial al realizar esta verificación radica en que el método 1, actualmente en uso en varios reglamentos de construcción (refs 3 y 4), podría llegar a sustituirse por el método 2.

Se han propuesto otros procedimientos para estimar Q (ref 5) que son función no lineal de los resultados del método 1; sin embargo, no se discuten en este trabajo porque han sido estudiados con base en estructuras sin amortiguamiento, las cuales, como se verá, conducen a conclusiones diferentes de las que corresponden a estructuras amortiguadas.

Para realizar estadísticamente este estudio, se emplearon técnicas de reducción de variancia del método de Monte Carlo.

En cuanto al análisis, este se limita a tres casos, los cuales se detallan en el Apéndice:

1. Torsión en estructuras de un piso, considerando que las respuestas dinámicas son la fuerza cortante y el momento torsionante.

2. Cabeceo en estructuras de un piso, considerando como respuestas la fuerza cortante y el momento de cabeceo.

3. Traslación en estructuras de dos pisos, tomando en cuenta las fuerzas cortantes en los entrepisos uno y dos.

2. CALCULO DE LAS RESPUESTAS MAXIMAS

Las respuestas elásticas máximas de los diversos tipos de estructuras se calcularon utilizando:

a) Método 1 (ec 1.1, criterio del Reglamento de Construcciones del Departamento del Distrito Federal, ref 3)

b) Método 2 (ec 1.2 y nuevo criterio de Rosenblueth, ref 2)

c) Análisis modal (respuesta exacta).

Los resultados del análisis modal sirvieron como base de comparación del grado de aproximación de las estimaciones logradas con los otros dos criterios.

Como excitaciones sísmicas se emplearon cuatro sismos simulados de acuerdo con el método indicado en la ref 6 (figs 1 a 4), y uno real (fig 5), registrado en la zona blanda de la ciudad de México (ref 7).

El análisis de los tres casos se realizó empleando el método de Monte Carlo, que consiste en estudiar el comportamiento de un modelo matemático determinado, mediante la simulación de los datos de entrada (generalmente en computadora digital) y del estudio estadístico de los resultados. Cada vez que se introduce un conjunto de datos y se obtiene la respuesta del modelo, se dice que se efectúa un *experimento conceptual* del problema; la colección de resultados constituye la *muestra* que sirve de base para inferir cuál es el grado de aproximación con que dicho modelo matemático representa el fenómeno para el cual se formuló.

Conforme aumenta el número de parámetros que intervienen en el modelo matemático, se incrementa la cantidad de experimentos necesaria para dilucidar cuáles influyen en el problema, es decir, para verificar si en los resultados que se obtienen al variar los valores de los parámetros existen diferencias estadísticas significativas; sin embargo, eso representa un costo de computación que en ocasiones hace prohibitivo tal tipo de estudios, a menos que se emplee alguna técnica de *reducción de variancia* (refs 11 y 12), lo que permite un ahorro considerable en el número de experimentos necesario para obtener conclusiones adecuadas.

La técnica de reducción de variancia que se emplea en este trabajo es muy común y consiste en:

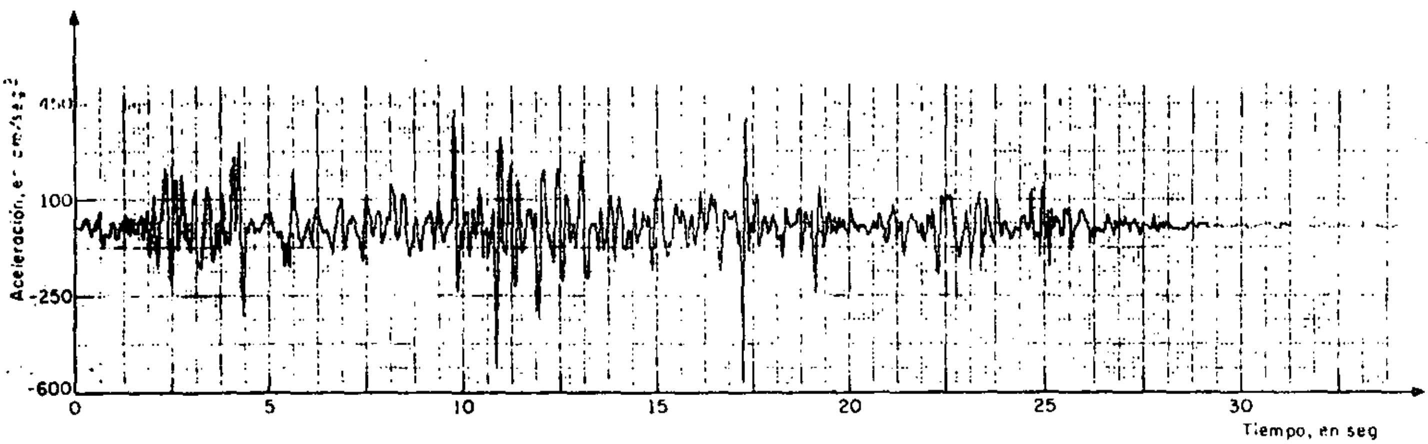


Fig 1. Sismo simulado No 1

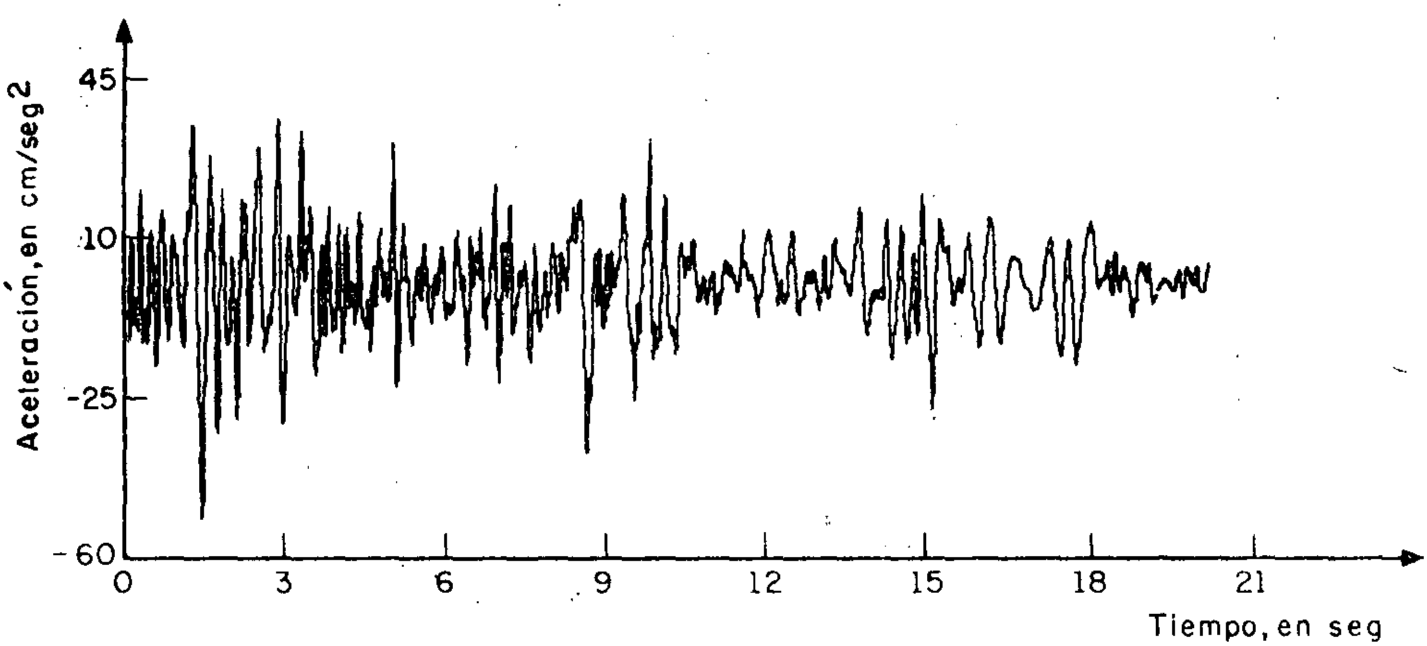
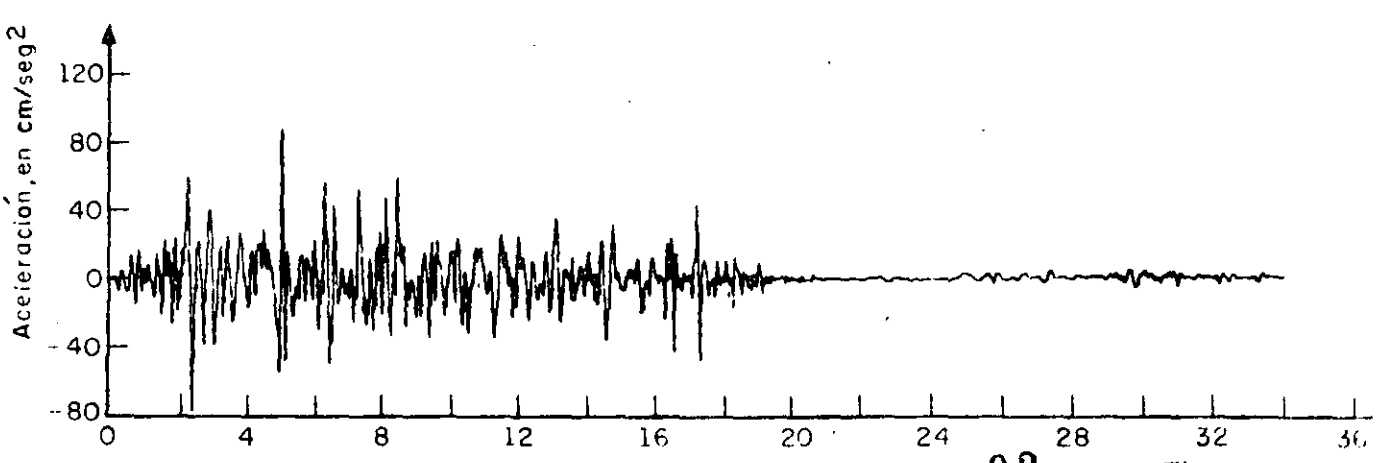


Fig 2. Sismo simulado No 2



92

Fig 3. Sismo simulado No 3

83-1

INSTITUTO DE INGENIERIA

ESTUDIO ESTADISTICO DE LOS CRITERIOS PARA ESTIMAR LA RESPUESTA SISMICA DE SISTEMAS LINEALES CON DOS GRADOS DE LIBERTAD

OCTAVIO A RASCON
AUGUSTO G VILLARREAL

OCTUBRE 1973 323

UNIVERSIDAD NACIONAL AUTONOMA DE MEXICO

Estudio estadístico de los criterios para estimar la respuesta sísmica de sistemas lineales con dos grados de libertad

Octavio A. Rascón
Augusto G. Villarreal*

RESUMEN

El objeto de este trabajo es verificar el grado de aproximación de dos métodos que con frecuencia se utilizan para estimar la respuesta sísmica máxima de sistemas lineales con varios grados de libertad. Para ello se aplica el método de Monte Carlo en el estudio de tres tipos de estructuras con dos grados de libertad: torsión y traslación, cabeceo y traslación, y traslación en dos pisos. Como excitaciones se utilizan sismos simulados y reales; se comparan las respuestas estimadas con las exactas, se hacen recomendaciones acerca del empleo de dichos métodos, y se obtienen las distribuciones de probabilidades de los cocientes de las respuestas exactas entre las estimadas.

ABSTRACT

The purpose of this work is to verify the degree of approximation of two methods used frequently for estimating the maximum seismic response of linear systems with various degrees of freedom. To do this, the Monte Carlo method is used in the study of three types of structures with two degrees of freedom: torsion and translation, rocking and translation, and translation in a two story building. Simulated and real earthquakes are used as ground excitations; estimated responses are compared with the exact ones, recommendations for the use of such methods are given, and the probability distributions of the ratios of exact to estimated responses are obtained.

1. INTRODUCCION

En este trabajo se analiza el comportamiento dinámico de algunos tipos de estructuras de comportamiento lineal de dos grados de libertad cuando se les sujeta a solicitaciones sísmicas. El objeto es verificar el grado de aproximación de dos métodos propuestos por Rosenblueth (refs 1 y 2) para estimar la respuesta máxima total, mediante su comparación con las respuestas máximas exactas obtenidas con el método de análisis modal, al superponer en el tiempo los efectos del sismo en los dos modos naturales de vibración de la estructura.

El método 1 consiste en estimar la respuesta máxima total, Q, extrayendo la raíz cuadrada de la suma de los cuadrados de la respuesta en cada modo natural de vibración, Q_i, es decir

$$Q = \sqrt{\sum_{i=1}^{n} Q_i^2} \qquad (1.1)$$

* Profesores investigadores, Instituto de Ingeniería, UNAM

PRIMER MODO

Parámetros	*Valores (2º ciclo)*		*Factor común*
x, ε	438	1	
F, M	9130	1,386,000	ω_1^2
F_0, M_0	9130	5,766,000	ω_1^2
x_0, ε_0	0.4860	0.00910	ω_1^2
α, β	0.7210	0.00187	ω_1^2
$\beta\delta, \alpha\theta$	0.3892	0.002585	ω_1^2
x_1, ε_1	1.1102	0.004455	ω_1^2
x_2, ε_2	4.365	———	ω_1^2
$x' \varepsilon'$	5.961	0.013565	ω_1^2
ω_1^2	73.5	75.8	—

Suponiendo que la aproximación es suficiente resulta

$$x'/\varepsilon' = 440, \ \bar{X}_1^T = [440,1], \ \omega_1^2 \doteq 74 \ (\text{rad/seg})^2$$

$$T_1 = 0.731 \text{ seg.}$$

El procedimiento para el cómputo de los parámetros del segundo modo es el mismo, sólo que la configuración supuesta deberá "limpiarse", antes de proseguir el cálculo, de las componentes del primer modo que pudiera contener. Se demuestra [7] que si $\bar{X}_2'$ es el vector de la configuración supuesta, el vector libre de componentes del primer modo queda dado por

$$X_2 = X_2' - \frac{\bar{X}_1^T \bar{M} \bar{X}_2'}{\bar{X}_1^T M \bar{X}_1} X_1 \qquad (32)$$

Suponiendo para el primer ciclo

$$\bar{X}_2' = \begin{bmatrix} -150 \\ 1 \end{bmatrix}$$

y sustituyendo valores en la ecuación matricial 32 se obtiene

$$\bar{X}_2 = \begin{bmatrix} -151 \\ 1 \end{bmatrix}$$

que nos da los valores de partida para el primer ciclo de cálculo.

SEGUNDO MODO

Parámetros	*Valores (1er. ciclo)*		*Factor común*
x, ε	—151	1	
F, M	—3143	1,386,000	ω_2^2
F_0, M_0	—3143	—123,000	ω_2^2
x_0, ε_0	—0.1672	—0.0001940	ω_2^2
α, β	—0.2481	0.0018700	ω_2^2
$\beta\delta, \alpha\theta$	0.3892	—0.0008890	ω_2^2
x_1, ε_1	0.1411	0.0009810	ω_2^2
x_2, ε_2	—0.0930	———	ω_2^2
x', ε'	—0.1191	0.0007870	ω_2^2
ω_2^2	1267	1270	—

$x'/\varepsilon' = -151, \ \bar{X}_2^T = [-151 \ 1], \ T_2 = 0.176$ seg.

En este caso se supuso un valor cercano al real y por tanto sólo se necesitó un ciclo para que se obtuviera la aproximación deseada. Si el valor supuesto no hubiese sido ese sino otro cualquiera seguramente no hubiera sido suficiente un ciclo de cálculo. En los ciclos subsiguientes se procedería en igual forma que antes: suponer inicialmente la configuración obtenida en el ciclo anterior; limpiarla de las componentes del primer modo; etc.

b) Respuesta sísmica

Los valores de los coeficientes de participación y de las ordenadas espectrales para este caso son:

$$C_1 = 0.001689, \qquad C_2 = -0.001689$$

$$S_{a1} = 127.4 \text{ cm/seg}^2, \qquad S_{a2} = 86.6 \text{ cm/seg}^2$$

Las respuestas máximas para cada modo valen

$$\begin{bmatrix} V_1 \\ M_1 \end{bmatrix} = \begin{bmatrix} 1{,}970 \text{ kg} \\ 298{,}200 \text{ kg cm} \end{bmatrix}$$

$$\begin{bmatrix} V_2 \\ M_2 \end{bmatrix} = \begin{bmatrix} 461 \text{ kg} \\ 203{,}000 \text{ kg cm} \end{bmatrix}$$

Las respuestas máximas totales serán (fig 10b)

$$V = 2{,}030 \text{ kg}$$

$$M = 361{,}000 \text{ kg cm}$$

$$M_b = 1{,}209{,}000 \text{ kg cm}$$

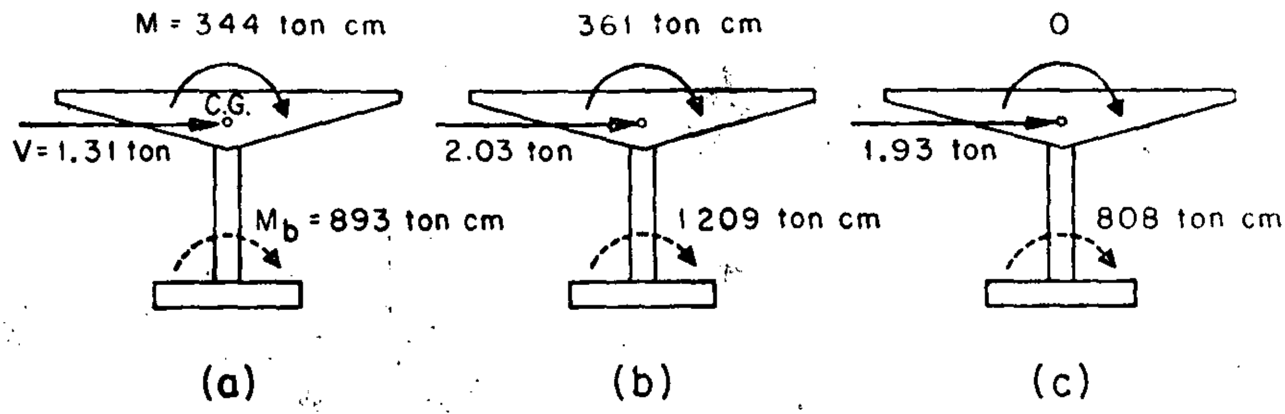


FIG. 10. *Respuestas sísmicas*

CASO 3. BASE RÍGIDA Y MASA CONCENTRADA

Para comparación de resultados se verá cuál es el valor de la respuesta máxima en el caso de despreciar la inercia rotacional y la interacción suelo-estructura.

Para este caso $p^2 = 608$ (rad/seg)^2, $T = 0.325$ seg, $0.15S_a = 92.6$ cm/seg^2, $V = mS_a = 1{,}930$ kg y $M_b = 808{,}000$ kg cm (fig 10c).

CONCLUSIONES

En la siguiente tabla se resumen los resultados de los tres casos, indicados como porcentajes del segundo caso.

Concepto	*Caso 1*	*Caso 2*	*Caso 3*
V	64.4%	100%	95.0%
M	95.2%	100%	0 %
M_b	73.8%	100%	66.7%

Los resultados de la tabla anterior dan una idea clara de la importancia que tiene el considerar la inercia rotacional de la cubierta y la interacción suelo-estructura. La importancia del primer concepto aumentará conforme mayor sea el momento de inercia de masa de la cubierta con respecto al eje z. El último concepto es tanto más importante cuanto más blando sea el suelo de cimentación. En particular puede observarse que en el tipo de solución 3 no se obtiene momento flexionante a la altura de C.G. Esto puede traer consigo serios errores en la cuantía del acero de refuerzo necesario en la unión columna-cubierta que es donde más ductilidad necesita desarrollarse.

AGRADECIMIENTO

El autor manifiesta su agradecimiento a los doctores E. Rosenblueth y J. A. Nieto, así como al Ing. E. del Valle por sus valiosos comentarios y sugerencias.

REFERENCIAS

1. McLean, R. S., "Inverted pendulum structures", technical report of *Consulting Civil and Structural Engineers*, Fullerton, Cal. (ene, 1965).
2. Barkan, D. D., "Dynamics of bases and foundations" McGraw Hill Book Co. Inc. (1962).
3. Jacobsen, L. S., y Ayre, R. S., "Engineering vibrations", McGraw Hill Book Co. Inc. (1958).
4. Newmark, N. M., "Numerical procedure for computing deflections, moments and buckling loads", *Transactions ASCE*, Vol. 108 (1943), pp. 1161-1234.
5. Rosenblueth, E. y Esteva, L., "*Proyecto de reglamento de las construcciones en el Distrito Federal. "Folleto complementario. Diseño sísmico de edificios*", Ediciones Ingeniería, México (1962).
6. Marsal, R., y Mazari, M., "*El subsuelo de la Ciudad de México*", Publicación del Instituto de Ingeniería, UNAM (1962).
7. Newmark, N. M., y Rosenblueth, E., "*Earthquake Engineering*", será publicado por Prentice-Hall, Inc.
8. Rosenblueth, E., "Some applications of probability theory in aseismic design", *Proceedings, 1st World Conference on Earthquake Engineering*, Berkeley, Cal. (1956), paper 8.

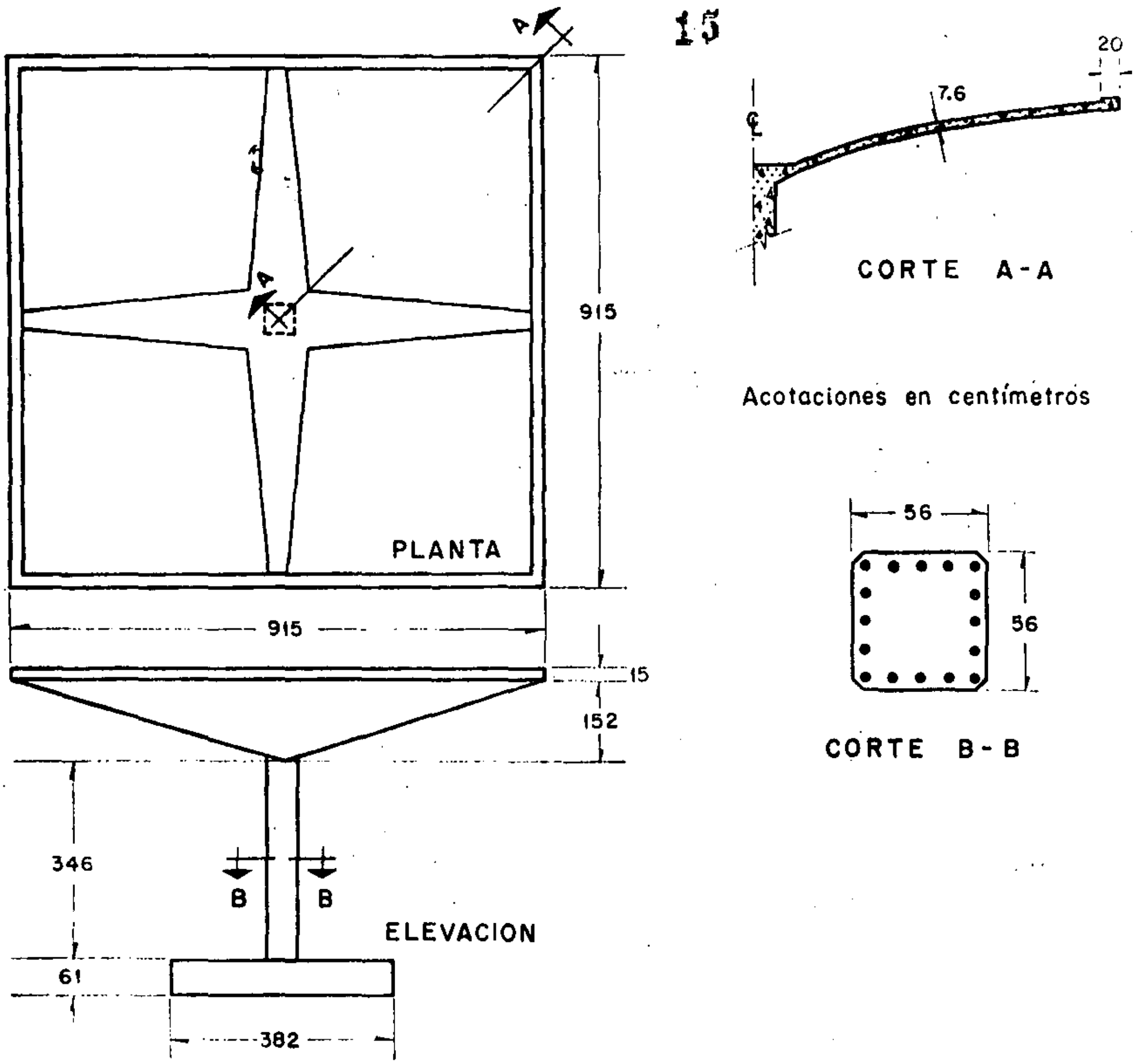


FIG. 8. *Cascarón utilizado para ejemplo.* (Después de R. McLean)

en la cual

$\bar{i}$ es un vector que representa los desplazamientos estáticos de cada grado de libertad de la estructura inducidos por un desplazamiento estático unitario de la base.

$\bar{X}_n$ es el vector modal para el enésimo modo (n)

$\bar{M}$ es la matriz de inercia y

$\bar{X}_n^T$ es el vector traspuesto de $\bar{X}_n$

Para nuestro caso se tendrá

$$\bar{i} = \begin{bmatrix} X_{est} \\ \varepsilon_{est} \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \end{bmatrix}$$

$$\bar{X}_1 = \begin{bmatrix} 238 \\ 1 \end{bmatrix}, \qquad \bar{X}_2 = \begin{bmatrix} -275 \\ 1 \end{bmatrix}$$

$$\bar{X}_1^T = \begin{bmatrix} 238 & 1 \end{bmatrix}, \qquad \bar{X}_2^T = \begin{bmatrix} -275 & 1 \end{bmatrix}$$

$$\bar{M} = \begin{bmatrix} m & 0 \\ 0 & J \end{bmatrix} = \begin{bmatrix} 20.81 & 0 \\ 0 & 1.386 \times 10^6 \end{bmatrix}$$

Sustituyendo valores en ec 27 y efectuando los productos matriciales en ella indicados se obtiene

$$C_1 = \frac{4,960}{2.566 \times 10^6} = 0.00193$$

$$C_2 = \frac{-5,720}{2.959 \times 10^6} = -0.00193$$

El valor absoluto de la respuesta máxima en cada uno de los modos será [7].

$$\begin{bmatrix} V_n = \text{fuerza cortante} \\ M_n = \text{momento flexionante} \end{bmatrix} = |C_n| \begin{bmatrix} m & 0 \\ 0 & J \end{bmatrix} \times$$

$$\times \begin{bmatrix} x_n \\ \varepsilon_n \end{bmatrix} S_{an} \tag{28}$$

donde

S_{an} = ordenada del espectro de aceleraciones afectada por el coeficiente sismico $C = 0.15$.

El espectro que será utilizado es el propuesto en el reglamento de construcciones del Distrito Federal [5] (fig. 9). Los valores de las ordenadas espectrales correspondientes a T_1 y T_2 son 100 cm/seg² y 80.6 cm/seg² respectivamente.

Sustituyendo valores en ec 28 se llega a

$$\begin{bmatrix} V_1 \\ M_1 \end{bmatrix} = \begin{bmatrix} 957 \text{ kg} \\ 268{,}000 \text{ kg cm} \end{bmatrix} \tag{29}$$

$$\begin{bmatrix} V_2 \\ M_2 \end{bmatrix} = \begin{bmatrix} 893 \text{ kg} \\ 216{,}000 \text{ kg cm} \end{bmatrix} \tag{30}$$

El criterio propuesto en ref. 8 será utilizado para el cálculo de la respuesta total (considerando los efectos combinados de los dos modos). Por lo anterior la respuesta total de la estructura valdrá

$$V = \sqrt{V_1^2 + V_2^2} \quad ; \quad M = \sqrt{M_1^2 + M_2^2} \tag{31a, 31b}$$

En ecs 31a y 31b

V = fuerza cortante total en la columna

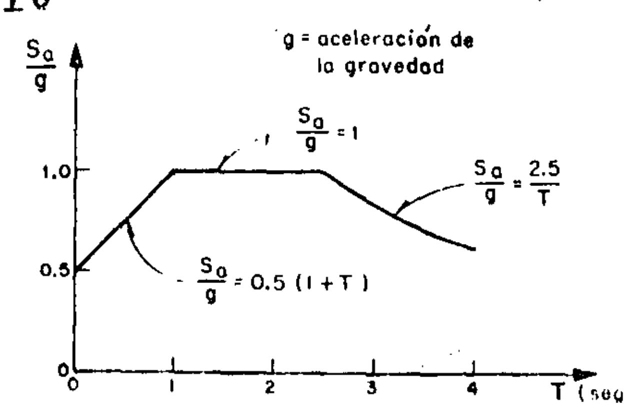


FIG. 9. *Espectro de aceleraciones*
(Después de E. Rosenblueth y L. Esteva)

M = momento flexionante total en C. G.

Sustituyendo los valores dados en ecs 29 y 30 en (31) se obtiene

$$V = 1{,}310 \text{ kg} \quad ; \quad M = 344{,}000 \text{ kg cm}$$

El momento en la base de la columna valdrá

$$M_b = 344{,}000 + 1{,}310 \times 419 = 893{,}000 \text{ kg cm}$$

Los resultados de este caso se resumen en la fig. 10a.

CASO 2. SUELO FLEXIBLE

a) *Cálculo de frecuencias y modos de vibración*

Para considerar las restricciones del suelo emplearemos el método propuesto anteriormente procediendo en forma tabular. Sustituyendo valores en ecuaciones para K y R se obtienen 1.88×10^4 kg/cm y 6.35×10^8 kg cm/rad respectivamente.

PRIMER MODO

Parámetros	*Valores (1er. ciclo)*		*Factor común*
x, ε (supuestos)	$x = 400$ cm	$\varepsilon = 1$ rad	
$F = m\,\omega_1^2 x,\ M = J\,\omega_1^2 \varepsilon$	$F = 8320$	$M = 1{,}386{,}000$	ω_1^2
$F_0 = F,\ M_0 = M + FL'$	$F_0 = 8320$	$M_0 = 5{,}376{,}000$	ω_1^2
$x_0 = F_0/K,\ \varepsilon_0 = M_0/R$	$x_0 = 0.4420$	$\varepsilon_0 = 0.00847$	ω_1^2
$\alpha = F/k,\ \beta = M/k,$	$\alpha = 0.6570$	$\beta = 0.00187$	ω_1^2
$\beta\,\delta,\ \alpha\,\theta$	$\beta\,\delta = 0.3892$	$\alpha\,\theta = 0.00235$	ω_1^2
$x_1 = \alpha + \beta\,\delta,\ \varepsilon_1 = \beta + \alpha\,\theta$	$x_1 = 1.0462$	$\varepsilon_1 = 0.00422$	ω_1^2
$x_2 = \varepsilon_0 L'$	$x_2 = 4.0650$	———	ω_1^2
$x' = x_0 + x_1 + x_2,\ \varepsilon' = \varepsilon_0 + \varepsilon_1$	$x' = 5.5532$	$\varepsilon' = 0.01269$	ω_1^2
$\omega_1^2 = x/x',\ \omega_1^2 = \varepsilon/\varepsilon'$	72.0	78.7	

$$x'/\varepsilon' = 438, \quad X_1^T = [438\ \ 1]$$

2. *Suelo flexible*

Al oscilar una estructura cimentada en suelo blando, existe interacción dinámica suelo-estructura que en la mayoría de los casos no debe despreciarse al calcular las frecuencias y los modos de vibración. En lo que sigue se propone la adaptación de un método numérico para tomar en cuenta dicho efecto.

Las restricciones del suelo serán idealizadas mediante resortes de comportamiento lineal: uno para desplazamientos lineales horizontales y otro para deformaciones angulares de cabeceo de la cimentación [2,3].

En la fig. 7 se hace referencia a los parámetros que a continuación se mencionan

$K =$ rigidez del resorte correspondiente a la traslación de la base [2] $= C_\tau A$

$C_\tau =$ coeficiente de cortante elástico uniforme del suelo.

$A =$ área de contacto de la cimentación.

$R =$ rigidez del resorte correspondiente a rotación de la base [2] $= C_\varphi I_b - W'\bar{y}$

$C_\varphi =$ coeficiente de compresión elástica no uniforme del suelo.

$I_b =$ momento de inercia de área de la base de la cimentación con respecto al eje z'

$W' =$ peso total de la estructura

$\bar{y} =$ altura del centro de gravedad de la estructura sobre el nivel de desplante

$F = m\omega_n^2 x$

$x =$ desplazamiento lineal total en C.G.

$M = J\omega_n^2 r$

$r =$ desplazamiento angular total en C.G.

$L' =$ altura de C.G. sobre el nivel de desplante

$x_0 =$ traslación de la base

$r_0 =$ rotación de la base

$x_1 = \alpha + \beta\delta$

$r_1 = \beta + \alpha\Theta$

$x_2 = L'r_0$

$\alpha = F/k$

$\beta = M/k_r$

J, L, δ, Θ, k, k_r, x_1, r_1 y W ya definidos anteriormente.

El problema será resuelto utilizando un procedimiento iterativo y la tabulación propuesta por N. M. Newmark[1]; se despreciarán la variación de la rigidez de la columna debida a la fuerza normal W y los momentos en la misma, causados por la excentricidad del peso debida a deformaciones de la columna.

Sean

$F_0 =$ fuerza horizontal en la base de la cimentación $= F$

$M_0 =$ momento flexionante en la base de la cimentación $= M + FL'$

$x_0 = F_0/K$

$r_0 = M_0/R$

A continuación se describe el procedimiento a seguir:

1. Suponer valores para x y r
2. Calcular F y M usando las expresiones $F = m\omega_n^2 x$ y $M = J\omega_n^2 r$. En esta etapa el valor de ω_n aún no se conoce; por tanto se llevará como factor común en el resto del cálculo

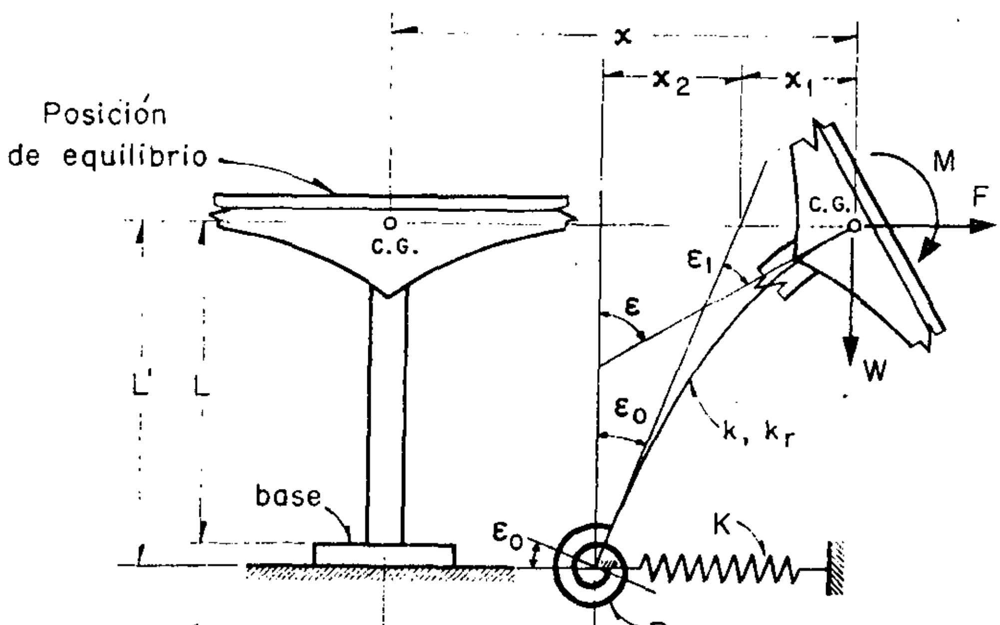


Fig. 7. *Modelo de interacción dinámica suelo-estructura*

3. Calcular la fuerza y el momento en la base mediante las fórmulas

$$F_0 = F \quad \text{y} \quad M_0 = M + FL'$$

4. Encontrar los valores de los desplazamientos $x_0 = F_0/K$ y $\varepsilon_0 = M_0/R$
5. Calcular los valores de los parámetros $\alpha = F/k$ y $\beta = M/k_r$
6. Efectuar los productos $\beta\delta$ y $\alpha\Theta$
7. Calcular $x_1 = \alpha + \beta\delta$ y $\varepsilon_1 = \beta + \alpha\Theta$
8. Efectuar el producto $x_2 = L'\varepsilon_0$
9. Calcular los desplazamientos lineales y angulares totales de C.G. mediante las expresiones

$$x' = x_0 + x_1 + x_2 \quad \text{y} \quad \varepsilon' = \varepsilon_0 + \varepsilon_1$$

10. Encontrar el valor de ω_n^2 mediante los cocientes x/x' y ε/ε'
11. Si los valores de ω_n^2 calculados en el paso anterior son aproximadamente iguales, el proceso habrá concluido. En caso contrario repítase la secuela utilizando como valores de partida para x y ε los encontrados en etapa 9 o valores cuyo cociente sea igual al de x' entre ε'. El proceso deberá continuarse hasta lograr la aproximación deseada.

EJEMPLO DE APLICACION

Con motivo de ilustrar los conceptos enunciados anteriormente se calcularán las frecuencias y modos de vibración de un cascarón ya construido en California, EUA (fig 8). Los datos necesarios han sido extraídos de la ref 1. Se computarán también las respuestas sísmicas suponiendo que esa estructura fuera a construirse en la zona blanda de la ciudad de México. Se utilizarán por tanto los parámetros elásticos de las arcillas del Valle de México y los espectros de diseño propuestos en el reglamento de construcción para el Distrito Federal.

Los datos necesarios de la estructura son

$L = 419$ cm
$L' = 480$ cm
$\bar{y} = 249$ cm
$W = 20.450$ kg ($m = 20.81$ kg seg^2/cm)
$W'' = 43.600$ kg
$I_0 = 1.775 \times 10^6$ cm^4
$I = 1.065 \times 10^6$ cm^4
$k = 1.266 \times 10^4$ kg/cm
$k_r = 7.41 \times 10^8$ kg cm/rad
$J = 1.386 \times 10^6$ kg seg^2 cm
$\Theta = 0.00358$ rad/cm
$\delta = 208$ cm/rad

Las expresiones para C_τ y C_φ son las siguientes

$$C_\tau = F_1 \frac{E'}{1-\nu^2}\frac{1}{\sqrt{A}}; \quad C_\varphi = F_2 \frac{E'}{1-\nu^2}\frac{1}{\sqrt{A}} \qquad (26)$$

En ecs 26

E' = módulo de elasticidad del suelo
ν = relación de Poisson del suelo
A = área de contacto de la cimentación
F_1, F_2 = factores de forma de la cimentación

Para el caso de la zona blanda del Valle de México un valor representativo de E' es 50 kg/cm^2 y $\nu \doteq 0.5$. Para una cimentación cuadrada los valores de F_1 y F_2 son 0.704 y 2.11 respectivamente.

Sustituyendo valores en ecs 26 se obtiene

$$C_\tau = 0.123 \ \text{kg/cm}^3$$
$$C_\varphi = 0.369 \ \text{kg/cm}^3$$

CASO 1. SUELO RÍGIDO

a) *Cálculo de frecuencias y modos de vibración*

Para el cálculo de las frecuencias de vibración usaremos la fórmula dada en ec 22. Los valores de los parámetros a sustituir son

$$p^2 = k/m = 608 \ (\text{rad/seg})^2$$
$$\Omega^2 = k_r/J = 535 \ (\text{rad/seg})^2$$
$$\mu = \Omega^2/p^2 = 0.882$$

con los cuales

$$\lambda_{1,2} = 2(1.882 \pm \sqrt{3.55 - 0.882}) = 0.494;\ 7.034$$

Por tanto

$$\omega_1 = \sqrt{0.494 \times 608} = \sqrt{300} = 17.32 \ \text{rad/seg}$$

$$\omega_2 = \sqrt{7.034 \times 608} = \sqrt{4260} = 65.30 \ \text{rad/seg}$$

Los períodos naturales son

$T_1 = 2\pi/\omega_1 = 0.362$ seg (T_1 obtenido de un registro de vibraciones libres de la estructura y reportado en ref 1 = 0.483 seg)

$T_2 = 2\pi/\omega_2 = 0.096$ seg

Comparando los valores calculado y medido de T_1 se puede ver la importancia de la interacción dinámica suelo-estructura.

Las relaciones modales se obtienen de las ecs. 25 y sus valores son

$$x_1/\varepsilon_1 = \frac{2 \times 419}{4 - 0.494} = 238 \ \text{cm/rad}$$

$$x_2/\varepsilon_2 = \frac{2 \times 419}{4 - 7.034} = -275 \ \text{cm/rad}$$

b) *Respuesta sísmica*

Para el cálculo de la respuesta sísmica de sistemas de varios grados de libertad es necesario calcular los coeficientes de participación de cada modo de vibración. Se puede demostrar que para este caso es aplicable la siguiente ecuación

$$C_n = \frac{\bar{X}_n^T \bar{M}\,\bar{I}}{\bar{X}_n^T \bar{M} \bar{X}_n} \qquad (27)$$

Despreciando las deformaciones por cortante, las expresiones para k, k_r, θ y δ pueden encontrarse por estática y valen

$$k = 3EI_c/L^3; \qquad (1a)$$

$$k_r = EI_c/L; \qquad (2a)$$

$$\theta = 1.5/L \qquad (1b)$$

$$\delta = L/2 \qquad (2b)$$

Para una fuerza de magnitud αk, el desplazamiento será α y el giro $\alpha\theta$. Para un par de magnitud βk_r el giro será β y el desplazamiento $\beta\delta$. Al aplicarse ambos simultáneamente, el desplazamiento total de C.G. será x_1 y el giro ε_1 (fig. 3).

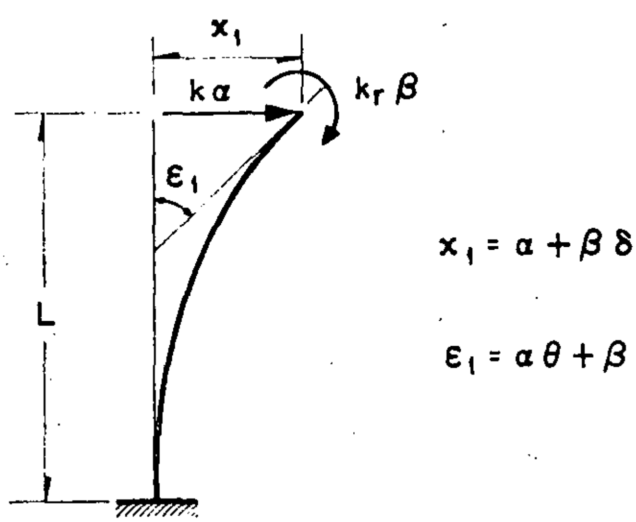


FIG. 3. *Desplazamientos y giros totales*

Por tanto los valores de x_1 y ε_1 quedan dados por

$$x_1 = \alpha + \beta\delta \qquad (3)$$

$$\varepsilon_1 = \alpha\theta + \beta \qquad (4)$$

Resolviendo el sistema de ecuaciones 3 y 4 para α y β, y utilizando las ecs $1b$ y $2b$ se obtiene

$$\alpha = (x_1 - k_r\gamma\varepsilon_1)/\kappa; \qquad (5a)$$

$$\beta = (\varepsilon_1 - k\gamma x_1)/\kappa \qquad (5b)$$

en las cuales

$$\gamma = L^2/2EI_c; \qquad (6a)$$

$$\kappa = 1 - kL^3/4EI_c = 0.25 \qquad (6b)$$

Para las oscilaciones del péndulo mostrado en la fig 1, el diagrama de cuerpo libre de la cubierta está indicado en la fig 4. Las ecuaciones de movimiento, despreciando efectos gravitacionales, serán

$$m\ddot{x}_1 + k\alpha = 0 \qquad (7)$$

$$J\ddot{\varepsilon}_1 + k_r\beta = 0 \qquad (8)$$

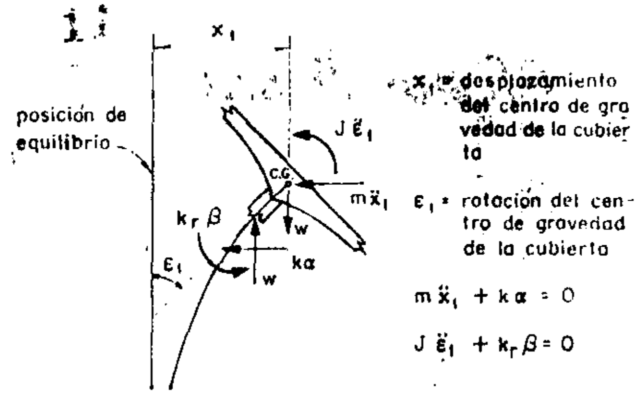


FIG. 4. *Diagrama de cuerpo libre*

Sustituyendo a (5a) y (5b) en (7) y (8) se obtiene

$$m\ddot{x}_1 + (kx_1 - kk_r\gamma\varepsilon_1)/\kappa = 0 \qquad (9)$$

$$J\ddot{\varepsilon}_1 + (k_r\varepsilon_1 - kk_r\gamma x_1)/\kappa = 0 \qquad (10)$$

Las ecs. 9 y 10 se pueden expresar matricialmente en la forma

$$\begin{bmatrix} m & 0 \\ 0 & J \end{bmatrix}\begin{bmatrix} \ddot{x}_1 \\ \ddot{\varepsilon}_1 \end{bmatrix} + \frac{1}{\kappa}\begin{bmatrix} k & -\gamma kk_r \\ -\gamma kk_r & k_r \end{bmatrix}\begin{bmatrix} x_1 \\ \varepsilon_1 \end{bmatrix} = 0 \qquad (11)$$

Utilizando las ecs 1a, 2a y 6a se encuentra que

$$\gamma kk_r = Lk/2 \qquad (12)$$

Puesto que el movimiento es armónico se tiene que

$$\ddot{x}_1 = -\omega^2 x_1 \quad \text{y} \quad \ddot{\varepsilon}_1 = -\omega^2\varepsilon_1 \qquad (13)$$

en donde ω es la frecuencia circular natural de vibración.

Sustituyendo las ecs. 12 y 13 en (11) se obtiene

$$-\begin{bmatrix} m & 0 \\ 0 & J \end{bmatrix}\omega^2\begin{bmatrix} x_1 \\ \varepsilon_1 \end{bmatrix} + \frac{1}{\kappa}\begin{bmatrix} k & -\dfrac{Lk}{2} \\ -\dfrac{Lk}{2} & k_r \end{bmatrix}\begin{bmatrix} x_1 \\ \varepsilon_1 \end{bmatrix} = 0 \qquad (14)$$

Factorizando en la ec. 14

$$\left\{\frac{1}{\kappa}\begin{bmatrix} k & -\dfrac{Lk}{2} \\ -\dfrac{Lk}{2} & k_r \end{bmatrix} - \omega^2\begin{bmatrix} m & 0 \\ 0 & J \end{bmatrix}\right\}\begin{bmatrix} x_1 \\ \varepsilon_1 \end{bmatrix} = 0 \qquad (15)$$

La ec 15 representa un sistema de ecuaciones homogéneas, el cual, para tener solución diferente de la trivial, necesita que su determinante sea nulo. Por tanto

$$\begin{vmatrix} \dfrac{k}{\kappa} - m\omega^2 & -\dfrac{Lk}{2\kappa} \\ -\dfrac{Lk}{2\kappa} & \dfrac{k_r}{\kappa} - J\omega^2 \end{vmatrix} = 0 \qquad (16)$$

Desarrollando el determinante se llega a

$$mJ\omega^4 - \frac{1}{\kappa}(kJ + mk_r)\omega^2 + $$
$$+ \frac{1}{4\kappa^2}(4kk_r - L^2k^2) = 0 \qquad (17)$$

Dividiendo ambos miembros entre mJ y considerando que $L^2k^2 = 3kk_r$ se obtiene

$$\omega^4 - \frac{kJ + mk_r}{mJ\kappa}\omega^2 + \frac{kk_r}{4mJ\kappa^2} = 0 \qquad (18)$$

que es una ecuación de segundo grado en ω^2, cuyas soluciones son

$$\omega^2_{1,2} = \frac{kJ + mk_r}{2mJ\kappa} \mp$$
$$\mp \sqrt{\frac{(kJ + mk_r)^2}{4m^2J^2\kappa^2} - \frac{kk_r}{4mJ\kappa^2}} \qquad (19)$$

Dividiendo numerador y denominador de (19) entre mJ

$$\omega^2_{1,2} = \frac{k/m + k_r/J}{2\kappa} \mp$$
$$\mp \frac{1}{2\kappa}\sqrt{(k/m + k_r/J)^2 - (k/m)(k_r/J)} \qquad (20)$$

Llamando a

$k/m = p^2 =$ cuadrado de la frecuencia circular natural por traslación

$k_r/J = \Omega^2 =$ cuadrado de la frecuencia circular natural por rotación

se obtiene

$$\omega^2_{1,2} = 2\left(p^2 + \Omega^2 \mp \sqrt{(p^2 + \Omega^2)^2 - p^2\Omega^2}\right) \qquad (21)$$

Dividiendo ambos miembros de (21) entre p^2 y haciendo $\omega^2/p^2 = \lambda$ y $\Omega^2/p^2 = \mu$ se llega a

$$\lambda_{1,2} = 2\left(1 + \mu \mp \sqrt{(1 + \mu)^2 - \mu}\right) \qquad (22)$$

Es interesante notar que si $J = 0$ (masa concentrada) de la ec 17 se obtiene $\omega^2 = k/m = p^2$.

Las configuraciones modales pueden obtenerse de cualquiera de las dos ecuaciones algebraicas contenidas en la ecuación matricial dada en ec 15. La primera de ellas es

$$\left(\frac{k}{\kappa} - m\omega_n^2\right)x_{1,n} - \frac{Lk}{2\kappa}r_{1,n} = 0 \qquad (23)$$

donde el indice n indica el número del modo y de la cual se obtiene

$$x_{1,n}/r_{1,n} = \frac{Lk}{2\kappa}\Big/\left(\frac{k}{\kappa} - m\omega_n^2\right) \qquad (24)$$

dividiendo numerador y denominador de (24) entre m y considerando que $\kappa = 0.25$, $k/m = p^2$ y que $\lambda_n = \omega_n^2/p^2$ se llega a

$$x_{1,n}/r_{1,n} = 2L/(4 - \lambda_n) \qquad (25)$$

Si se desean tomar en cuenta las deformaciones por cortante basta con modificar las rigideces mediante un análisis de estática y partir de nuevo de la ec 17 sin considerar que $L^2k^2 = 3kk_r$. Si existe excentricidad en alguna dirección su efecto podrá tomarse en cuenta introduciendo un grado de libertad adicional.

En las figs 5 y 6 se encuentran representados los resultados de las ecs 22 y 25.

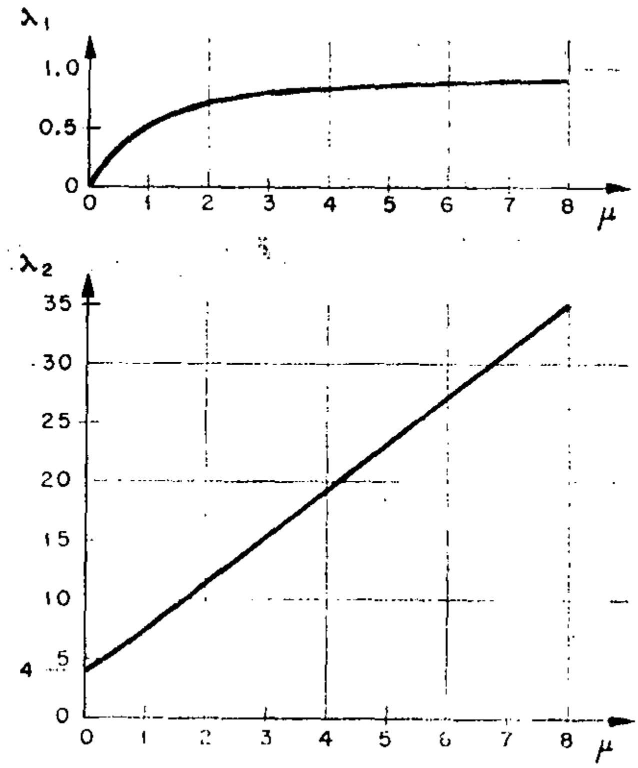


FIG. 5. *Gráfica de frecuencias*

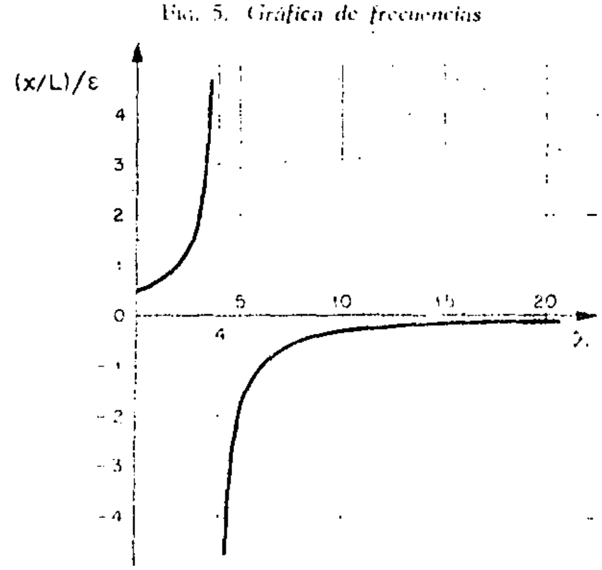


FIG. 6. *Gráfica* $(x/L)/\varepsilon$ vs λ

SUSTITUYENDO A ω_1^2, EN (1') O EN (2'):

$$\underline{z}_1 = \begin{bmatrix} z_1 \\ \\ \theta_1 \end{bmatrix} = \begin{bmatrix} 1 \\ \\ \dfrac{1 - \lambda_1^2}{cb} \end{bmatrix} ;$$

SUSTITUYENDO A ω_2^2:

$$\underline{z}_2 = \begin{bmatrix} z_2 \\ \\ \theta_2 \end{bmatrix} = \begin{bmatrix} 1 \\ \\ \dfrac{1 - \lambda_2^2}{cb} \end{bmatrix} \quad \text{o:} \quad \underline{z}_n = \begin{bmatrix} 1 \\ \\ \dfrac{1 - \lambda_n^2}{cb} \end{bmatrix}$$

Efectos sísmicos en estructuras en forma de péndulo invertido

Octavio RASCON CH. *

INTRODUCCION

En la práctica se presentan estructuras constituidas por una sola columna la cual sostiene una cubierta que puede ser una losa o un cascarón. Su comportamiento dinámico debe estudiarse considerando el efecto que la inercia rotacional de la cubierta induce en el movimiento total de la estructura.

A principios de este año se presentó en California, EUA, un trabajo[1] en el cual se trató este problema desde un punto de vista energético. Se calculó sólo el periodo fundamental y con base en él, la respuesta de la estructura a un determinado temblor. Los periodos calculados para cuatro estructuras de este tipo ya construidas fueron menores que los medidos *in situ*. La discrepancia fue atribuida a efectos de rotación y traslación de la base.

El objeto de este trabajo es introducir un análisis modal, el cual nos proporcionará los efectos del acoplamiento que existe entre los modos de vibración. También se tomarán en cuenta en forma aproximada los efectos de rotación y traslación de la base.

CALCULO DE FRECUENCIAS Y CONFIGURACIONES MODALES DE VIBRACION

1. *Suelo rígido*

Para el caso en que el centro de gravedad de la cubierta se encuentra localizado en la prolongación del eje de la columna, el movimiento de la estructura podrá estudiarse en dos direcciones perpendiculares entre sí. En tal caso el problema podrá discretizarse como de dos modos de vibración acoplados en cada dirección.

Para el cálculo de las frecuencias de vibración se idealizará la estructura como de comportamiento lineal, constituida por una cubierta infinitamente rigida de masa simétricamente distribuida y soportada por una sola columna. Como primer caso se considerará al suelo infinitamente rigido (fig. 1).

En fig 1

W = peso de la cubierta más la parte tributaria de la columna

J = momento de inercia de la masa de la cubierta respecto al eje z

* Asistente de Investigador, Instituto de Ingeniería, UNAM.

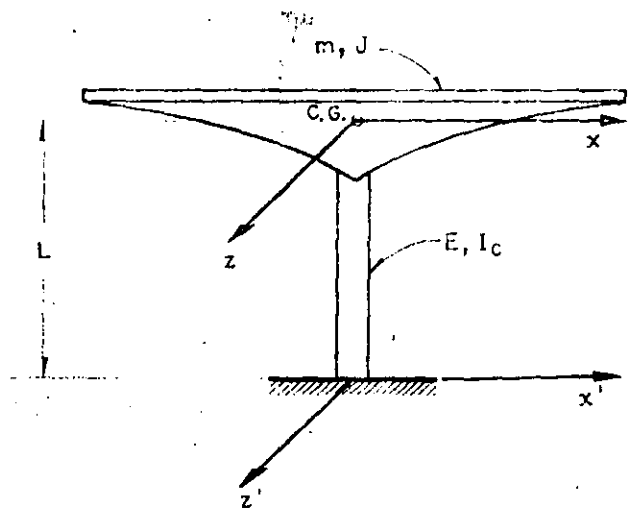


FIG. 1. *Péndulo invertido*

E = módulo de elasticidad del material de la columna

I_c = momento de inercia de la sección transversal de la columna con respecto al eje z

C.G. = centro de gravedad de la cubierta

L = distancia de C.G. al suelo.

Para la columna mostrada en las figs. 2a y 2b.

k = rigidez por traslación (fuerza horizontal aplicada en C.G. necesaria para que este se desplace la unidad)

k_r = rigidez por rotación (par aplicado en C.G. necesario para producir un giro unitario a la altura de C.G.

θ = rotación en C.G. debida a la fuerza k

δ = desplazamiento lateral de C.G. debido al momento k_r.

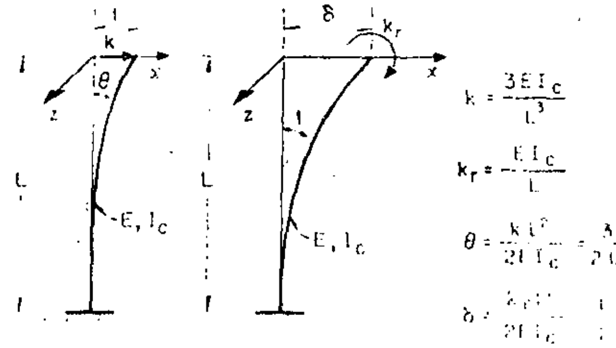
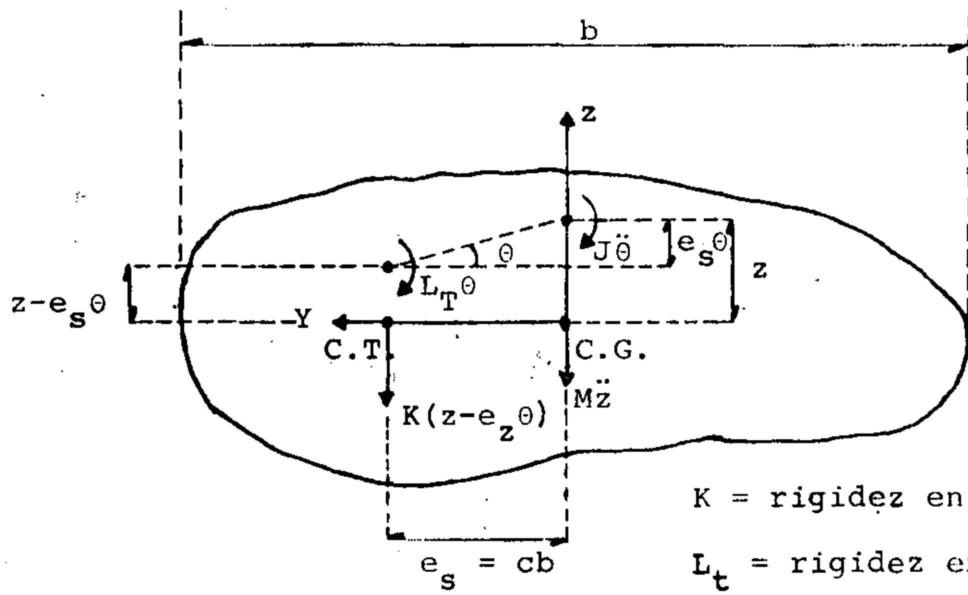


FIG. 2. *Rigideces*

PROBLEMA DE VIBRACIONES DE TORSION

ACOPLADA CON TRASLACION



K = rigidez en traslación

L_t = rigidez en torsión

$$\Sigma F_z = M\ddot{z} + K(z-e_s\Theta) = 0 \tag{1}$$

$$\Sigma M_{C.G.} = J\ddot{\Theta} + L_t\Theta - K(z-e_s\Theta)\ e_s = 0$$

$$J\ddot{\Theta} + L\Theta - Ke_s z = 0 \tag{2}$$

EN DONDE $L = L_t + Ke_s^2$

PUESTO QUE LAS VIBRACIONES SON ARMONICAS:

$\ddot{\Theta} = -\omega^2\Theta$ y $\ddot{z} = -\omega^2 z$

Sustituyendo en ec (1):

$-\omega^2 Mz + Kz - Ke_s\Theta = 0$

$$(K - \omega^2 M)z - Ke_s\Theta = 0 \tag{1'}$$

Sustituyendo (3) en (2):

$$-J\omega^2\theta + L_T\theta - Ke_s z = 0$$

$$(L_T - J\omega^2)\theta - Ke_s z = 0 \qquad (2')$$

$$\text{Det}\begin{bmatrix} K - \omega^2 M & -Ke_s \\ -Ke_s & L_T - J\omega^2 \end{bmatrix} = 0$$

$$(K - \omega^2 M)(L_T - J\omega^2) - K^2 e_s^2 = 0$$

$$KL_T - KJ\omega^2 - \omega^2 ML_T + MJ\omega^4 - K^2 e_s^2 = 0$$

$$\omega^4 - \frac{KJ + ML_T}{MJ}\omega^2 + \frac{KL_T}{MJ} - \frac{K^2 e_s^2}{MJ} = 0$$

DIVIENDO POR $(K/M)^2$:

$$\frac{\omega^4}{(K/M)^2} - \frac{\omega^2}{K/M}\;\frac{KJ + ML_T}{(MJ)(K/M)} + \frac{KL_T}{MJ(K/M)^2} - \frac{K^2 e_s^2}{MJ(K/M)^2} = 0$$

SI $\lambda^2 = \omega^2/(K/M)$ Y CONSIDERANDO $e_s = cb$:

$$\lambda^4 - \lambda^2\left(1 + \frac{L_T/J}{K/M}\right) + \frac{L_T/J}{K/M} - \frac{c^2}{J/(Mb^2)} = 0$$

SI $(L_T/J)/(K/M) = \eta$ y $j^2 = J/(Mb^2)$

$$\lambda^4 - \lambda^2(1 + \eta) + \eta - c^2/j^2 = 0$$

$$\therefore\ \lambda_{1,2}^2 = \frac{\eta + 1}{2} \pm \sqrt{\frac{(\eta + 1)^2}{4} + \frac{c^2}{j^2}}$$

$$\Rightarrow \omega_1^2 = \lambda_1 (K/M) \text{ y } \omega_2^2 = \lambda_2 (K/M)$$

Tomado del libro de N. Newmark y E. Rosenblueth D.

TABLA 2.1. *Ejemplo 2.7*

t seg	Q_1 ton	$\ddot{x}$ cm/seg²	$\dot{x}$ cm/seg	x_1 cm	$x_1 - x_0$ cm	Q_2 ton	$\ddot{y}$ cm/seg²	$\dot{y}$ cm/seg	x_2 cm	$x_2 - x_0$ cm	x_0 cm
0	0	0	0	0	0	0	0	0	0	0	0
0.2	2.540	1.350	0.135	0.0095	-0.2310	1.380	1.500	0.150	0.0100	-0.2300	0.24
0.2	2.546	1.270	0.127	0.0085	-0.2315	1.386	1.380	0.138	0.0092	-0.2308	0.24
0.2	2.546	1.273	0.127	0.0085	-0.2315	1.386	1.386	0.138	0.0092	-0.2308	0.24
0.4	4.549	+2.300	0.484	+0.0662	-0.4138	2.453	2.100	0.486	+0.0695	-0.4107	0.48
0.4	4.548	2.274	0.481	0.0660	-0.4140	2.455	2.468	0.523	0.0718	-0.4082	0.48
0.4	4.549	2.274	0.491	0.0660	-0.4140	2.455	2.455	0.522	0.0717	-0.4083	0.48
0.4	4.548	2.274	0.491	0.0660	-0.4140	2.455	2.455	0.522	0.0717	-0.4083	0.48
0.6	5.585	2.700	0.978	0.2105	-0.5095	2.960	3.200	1.088	0.2301	-0.4899	0.72
0.6	5.581	2.793	0.987	0.2111	-0.5089	2.967	2.960	1.064	0.2285	-0.4915	0.72
0.6	5.580	2.790	0.997	0.2111	-0.5089	2.966	2.967	1.065	0.2296	-0.4914	0.72
0.6	5.580	2.790	0.987	0.2111	-0.5089	2.966	2.966	1.063	0.2296	-0.4914	0.72
0.8	5.409	2.900	1.556	0.4550	-0.4950	2.790	2.980	1.660	0.5010	-0.4590	0.96
0.8	5.423	2.704	1.536	0.4537	-0.4963	2.798	2.790	1.641	0.4997	-0.4603	0.96
0.8	5.422	2.711	1.537	0.4638	-0.4962	2.797	2.798	1.642	0.4998	-0.4602	0.96
0.8	5.422	2.711	1.537	0.4638	-0.4962	2.797	2.797	1.642	0.4998	-0.4602	0.96
1.0	4.104	2.150	2.023	0.8216	-0.3784	1.977	2.200	2.142	0.8802	-0.3198	1.20
1.0	4.111	2.052	2.013	0.8210	-0.3790	1.985	1.977	2.120	0.8787	-0.3213	1.20
1.0	4.111	2.055	2.014	0.8210	-0.3790	1.985	1.985	2.121	0.8787	-0.3213	1.20
1.0	4.111	2.055	2.014	0.8210	-0.3790	1.985	1.985	2.121	0.8787	-0.3213	1.20
1.2	1.931	0.950	2.315	1.2575	-0.1825	0.712	0.700	2.390	1.3341	-0.1059	1.44
1.2	1.930	0.965	2.316	1.2576	-0.1824	0.712	0.712	2.391	1.3341	-0.1059	1.44
1.2	1.930	0.965	2.316	1.2576	-0.1824	0.712	0.712	2.391	1.3341	-0.1059	1.44
1.4	-0.643	0.320	2.381	1.7316	0.0516	-0.735	-0.800	2.382	1.8165	0.1365	1.68
1.4	-0.652	-0.326	2.380	1.7315	0.0515	-0.735	-0.735	2.388	1.8169	0.1369	1.68
1.4	-0.652	-0.326	2.380	1.7315	0.0515	-0.735	-0.735	2.388	1.8169	0.1369	1.68
1.6	-3.093	-1.500	2.197	2.1932	0.2732	-2.026	-2.100	2.104	2.2707	0.3507	1.92
1.6	-3.090	-1.541	2.193	2.1929	0.2729	-2.029	-2.026	2.111	2.2712	0.3512	1.92
1.6	-3.060	-1.540	2.193	2.1929	0.2729	2.029	-2.029	2.111	2.2712	0.3512	1.92
1.8	-4.830	-2.500	1.789	2.5943	0.4343	-2.869	-2.900	1.618	2.6471	0.4471	2.16
1.8	-4.936	-2.415	1.797	2.5949	0.4349	-2.871	-2.869	1.621	2.6473	0.4973	2.16
1.8	-4.936	-2.418	1.797	2.5949	0.4349	-2.871	-2.871	1.621	2.6473	0.4873	2.16
2.0	-5.547	-2.800	1.275	2.9034	0.5034	-3.069	-3.000	1.034	2.9132	0.5132	2.40
2.0	-5.549	-2.773	1.278	2.9036	0.5036	-3.068	-3.069	1.027	2.9127	0.5127	2.40
2.0	-5.549	-2.774	1.278	2.9036	0.5036	3.068	-3.068	1.027	2.9127	0.5127	2.40

TABLA 2.1.. *Ejemplo 2.7 (Cont.)*

t seg	Q_1 ton	$\ddot{x}_1$ cm/seg²	$\dot{x}_1$ cm/seg	x_1 cm	$x_1 - x_0$ cm	Q_2 ton	$\ddot{x}_2$ cm/seg²	$\dot{x}_2$ cm/seg	x_2 cm	$x_2 - x_0$ cm	x_0 cm
2.2	-10.156	-5.200	0.481	3.0875	0.9275	-5.332	-5.460	0.174	3.0408	0.8808	2.16
2.2	-10.165	-5.078	0.493	3.0883	0.9283	-5.337	-5.332	0.187	3.0417	0.8817	2.16
2.2	-10.165	-5.083	0.493	3.0883	0.9283	-5.337	-5.337	0.186	3.0417	0.8817	2.16
2.4	-12.578	-6.900	-0.705	3.0731	1.1531	6.386	-6.200	-0.968	2.9665	1.0465	1.92
2.4	-12.617	-6.209	-0.644	3.0772	1.1572	6.383	-6.386	-0.987	2.9652	1.0452	1.92
2.4	-12.615	-6.209	-0.646	3.0770	1.1570	-6.383	-6.383	-0.986	2.9652	1.0452	1.92
2.4	-12.615	-6.208	-0.646	3.0770	1.1570	-6.383	-6.383	-0.986	2.9652	1.0452	1.92
2.6	-12.388	-6.200	-1.697	2.8225	1.1425	-5.958	-6.000	-2.224	2.6429	0.9629	1.68
2.6	-12.388	-6.194	-1.696	2.8225	1.1425	-5.959	-5.958	-2.220	2.6432	0.9632	1.68
2.6	-12.388	-6.194	-1.696	2.8225	1.1425	-5.959	-5.959	-2.220	2.6432	0.9632	1.68
2.8	-9.573	-4.300	-2.945	2.3320	0.8920	-4.155	-4.100	-3.206	+2.0975	0.6575	1.44
2.8	-9.540	-4.767	-2.994	2.3288	0.8888	-4.150	-4.155	-3.212	2.0921	0.6521	1.44
2.8	-9.541	-4.770	-2.992	2.3289	0.8889	-4.150	-4.150	-3.211	2.0921	0.6521	1.44
2.8	-9.541	-4.770	-2.992	2.3289	0.8889	-4.150	-4.150	-3.211	2.0921	0.6521	1.44
3.0	-4.687	-2.500	-3.719	1.6502	0.4502	-1.376	-1.400	-3.766	1.3853	0.1853	1.20
3.0	-4.698	-2.343	-3.703	1.6513	0.4513	-1.378	-1.376	-3.764	1.3854	0.1854	1.20
3.0	-4.698	-2.349	-3.704	1.6513	0.4513	-1.378	-1.378	-3.764	1.3854	0.1854	1.20
3.2	1.090	0.500	-3.859	0.8545	-0.0755	1.748	1.700	-3.732	0.6255	-0.3345	0.96
3.2	1.106	0.545	-3.884	0.8620	-0.0772	1.748	1.748	-3.727	0.6259	-0.3341	0.96
3.2	1.105	0.553	-3.883	0.8629	-0.0771	1.748	1.748	-3.727	0.6259	-0.3341	0.96
3.2	1.105	0.553	-3.883	0.8629	-0.0771	1.748	1.748	-3.727	0.6259	-0.3341	0.96
3.4	6.608	3.600	-3.468	0.1377	-0.5823	4.506	4.700	-3.082	-0.0649	-0.7849	0.72
3.4	6.629	3.304	-3.428	0.1357	-0.5843	4.515	4.506	-3.101	-0.0662	-0.7862	0.72
3.4	[illegible]	3.314	-3.429	0.1356	-0.5842	4.515	4.515	-3.100	-0.0661	-0.7861	0.72
3.4	[illegible]	3.314	-3.439	0.1358	-0.5842	4.515	4.515	-3.100	-0.0661	-0.7861	0.72
3.6	10.578	5.400	-2.568	-0.4718	-0.9518	6.251	6.400	-1.958	-0.5799	-1.0599	0.48
3.6	10.589	5.269	-2.579	-0.4725	-0.9525	6.277	6.251	-2.023	-0.5842	-1.0642	0.48
3.6	10.589	5.259	-2.577	-0.4725	-0.9525	6.277	6.277	-2.020	-0.5841	-1.0641	0.48
3.6	10.589	5.259	-2.577	-0.4725	-0.9525	6.277	6.277	-2.020	-0.5841	-1.0641	0.48
3.8	12.259	6.200	-1.427	-0.6760	-1.1160	6.612	6.800	-0.712	-0.8591	-1.0991	0.24
3.8	12.264	6.110	-1.434	-0.6764	-1.1164	6.618	6.612	-0.731	-0.8603	-1.1003	0.24
3.8	12.264	6.132	-1.434	-0.6764	-1.1164	6.618	6.618	-0.730	-0.8603	-1.1003	0.24
4.0	11.323	5.600	-0.260	-1.0441	-1.0441	5.454	5.400	0.472	-0.8821	-0.8821	0
4.0	11.319	5.661	-0.255	-1.0437	-1.0437	5.453	5.414	0.477	-0.8817	-0.8817	0
4.0	11.319	5.660	-0.255	-1.0437	-1.0437	5.453	5.453	0.477	-0.8817	-0.8817	0
4.2	10.705	5.350	0.846	-0.9536	-0.9536	5.330	5.300	1.549	-0.869	-0.8691	0
4.2	10.705	5.352	0.846	-0.9536	-0.9536	5.329	5.330	1.552	-0.9689	-0.9689	0

EN $t = 0$, $y_i = x_i = 0$, $\dot{y}_i = \dot{x}_i = 0$, $\ddot{y}_i = \ddot{x}_i = 0.$

EN $t = 0.2$, $x_o = 1.2 \times 0.2 = 0.24$ cm; SUPONGAMOS $\ddot{x}_1 = \ddot{y}_1 = 1.35$

Y $\ddot{x}_2 = \ddot{y}_2 = 1.50$ cm/seg;

PRIMER CICLO

PARA LA MASA 1:

$$\dot{x}_1 = 0 + 0.1\ (0 + 1.35) = 0.135 \text{ cm/seg}$$

$$x_1 = 0 + 0 + 0.04(0 + 1.35/6) = 0.009 \text{ cm}$$

$$y_1 = 0.009 - 0.24 = -\ 0.231 \text{ cm}$$

PARA LA MASA 2:

$$\dot{x}_2 = 0 + 0.1(0 + 1.50) = 0.15$$

$$x_2 = 0 + 0 + 0.04(0 + 1.50/6) = 0.01$$

$$y_2 = 0.01 - 0.24 = -\ 0.23 \text{ cm}$$

$$\underline{Q} = \begin{bmatrix} Q_1 \\ Q_2 \end{bmatrix} = \begin{bmatrix} 10 & 1 \\ 1 & 5 \end{bmatrix} \begin{bmatrix} -0.231 \\ -0.230 \end{bmatrix} = \begin{bmatrix} -2.540 \\ -1.381 \end{bmatrix}$$

POR LO QUE

$$\ddot{y}_1 = \ddot{x}_1 = 2.54/2 = 1.27 \neq 1.35$$

$$\ddot{y}_2 = \ddot{x}_2 = 1.381/1 = 1.381 \neq 1.50$$

SEGUNDO CICLO

$$\dot{x}_1 = 0.1 \times 1.27 = 0.127$$

$$x_1 = 0.04 \times 1.27/6 = 0.0085$$

$$y_1 = 0.0085 - 0.24 = -0.2315$$

$$\dot{x}_2 = 0.1 \times 1.381 = 0.138$$

$$x_2 = 0.04 \times 1.381/6 = 0.0092$$

$$y_2 = 0.0092 - 0.24 = -0.2308$$

$$Q = \begin{bmatrix} 10 & 1 \\ 1 & 5 \end{bmatrix} \begin{bmatrix} -0.2315 \\ -0.2308 \end{bmatrix} = \begin{bmatrix} -2.546 \\ -1.386 \end{bmatrix}$$

DE DONDE $\ddot{x}_1 = \ddot{y}_1 = 2.546/2 = 1.273 \doteq 1.27$

$\ddot{x}_2 = \ddot{y}_2 = 1.386/1 = 1.386 \doteq 1.381$

EN $t = 0.2 + 0.2 = 0.4$ seg SE TIENEN $x_o = 1.2 \times 0.4 = 0.48$,

$x_1(t_i) = 0.0085 \quad ; \quad x_2(t_i) = 0.0092$

$\dot{x}_1(t_i) = 0.127 \quad ; \quad \dot{x}_2(t_i) = 0.138$

$\ddot{x}_1(t_i) = 1.273 \quad ; \quad \ddot{x}_2(t_i) = 1.386$

<u>PRIMER CICLO</u>

SUPONIENDO $\ddot{x}_1(t_{i+1}) = 2.3$ Y $\ddot{x}_2(t_{i+1}) = 2.1$ SE OBTIENEN:

$\dot{x}_1 = 0.127 + 0.1(1.273 + 2.3) = 0.484$

$x_1 = 0.0085 + 0.2 \times 0.127 + 0.04(1.273/3 + 2.3/6) = 0.0662$

$y_1 = 0.0662 - 0.48 = -0.4138$

$\dot{x}_2 = 0.138 + 0.1(1.386 + 2.1) = 0.486$

$x_2 = 0.0092 + 0.2 \times 0.138 + 0.04(1.386/3 + 2.1/6) = 0.0693$

$y_2 = 0.0693 - 0.48 = -0.4107$

$$Q = \begin{bmatrix} 10 & 1 \\ 1 & 5 \end{bmatrix} \begin{bmatrix} -0.4138 \\ -.4107 \end{bmatrix} = \begin{bmatrix} -4.548 \\ -2.468 \end{bmatrix}$$

DE DONDE $\ddot{x}_1 = \ddot{y}_1 = 4.548/2 = 2.274 \neq 2.3$

$\ddot{x}_2 = \ddot{y}_2 = 2.468 \neq 2.1$

ETCETERA. LOS RESULTADOS DEL PROBLEMA SE PRESENTAN EN LA TABLA I.

METODO β DE NEWMARK

SISTEMAS ELASTICOS LINEALES DE VARIOS GRADOS DE LIBERTAD

PARA CALCULAR LA RESPUESTA DE UN SISTEMA DE N GRADOS DE LIBERTAD Y COMPORTAMIENTO ELASTICO LINEAL SE EMPLEAN LAS MISMAS ECUACIONES QUE PARA UN SISTEMA DE UN GRADO DE LIBERTAD.

$$\dot{x}_j(t_{i+1}) = \dot{x}_j(t_i) + [\ddot{x}_j(t_i) + \ddot{x}_j(t_{i+1})] \frac{\Delta t}{2}$$

$$x_j(t_{i+1}) = x_j(t_i) + \dot{x}_j(t_i)\Delta t + [(1/2-\beta)\ddot{x}_j(t_i) + \beta\ddot{x}_j(t_{i+1})](\Delta t)^2$$

EN DONDE $j = 1,2,\ldots,N$.

EN ESTE CASO SE RECOMIENDA TAMBIEN UN VALOR DE β COMPRENDIDO ENTRE 1/4 Y 1/6, Y QUE $\Delta t \doteq 0.1\ T_N$, EN DONDE T_N ES EL PERIODO NATURAL DE VIBRACION MAS PEQUEÑO.

EJEMPLO

SEA UN SISTEMA DE DOS GRADOS DE LIBERTAD CON AMORTIGUAMIENTO NULO, CUYAS MATRICES DE MASAS Y RIGIDECES SON:

$$\underline{K} = \begin{bmatrix} 10 & 1 \\ 1 & 5 \end{bmatrix} \quad , \qquad \underline{M} = \begin{bmatrix} 2 & 0 \\ 0 & 1 \end{bmatrix}$$

USANDO EL METODO β DE NEWMARK CON $\Delta t=0.2$ seg Y $\beta= 1/6$ CALCULE LA RESPUESTA DINAMICA ANTE UNA EXCITACION DADA POR LOS DESPLAZAMIENTOS DEL SUELO:

$x_o = 1.2\ t$ SI $0 \leq t \leq 2$ seg (x_o EN CENTIMETROS)

$x_o = 4.8-1.2\ t$ SI $2 \leq t \leq 4$ seg

$x_o = 0$ SI $t < 0$ o $t > 4$ seg

PUESTO QUE ESTA EXCITACION IMPLICA QUE $\ddot{x}_o(t) = 0$ PARA TODO t, SE TIENE QUE LA ECUACION MATRICIAL DE EQUILIBRIO RESULTA SER

$$\underline{M}\ddot{\underline{Y}} + \underline{K}\underline{Y} = \underline{M}\ddot{\underline{Y}} + \underline{Q} = \underline{0}$$

POR LO QUE

$$m_1 \ddot{y}_1 + Q_1 = 0 \quad \rightarrow \quad \ddot{y}_1 = Q_1/m_1$$

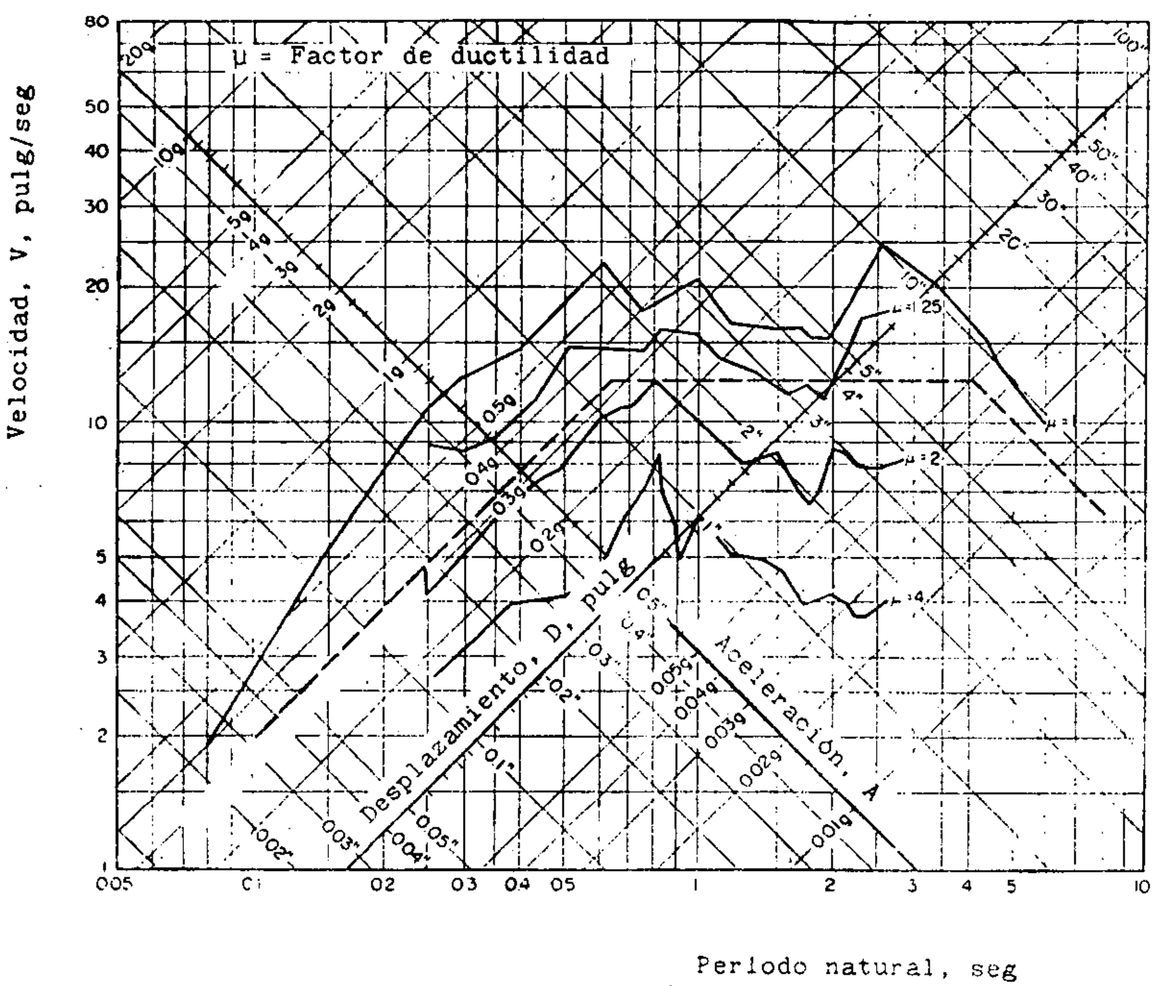
$$m_2 \ddot{y}_2 + Q_2 = 0 \quad \rightarrow \quad \ddot{y}_2 = Q_2/m_2$$

EN DONDE $y_1 = x_1 - x_o$ Y $y_2 = x_2 - x_o$.

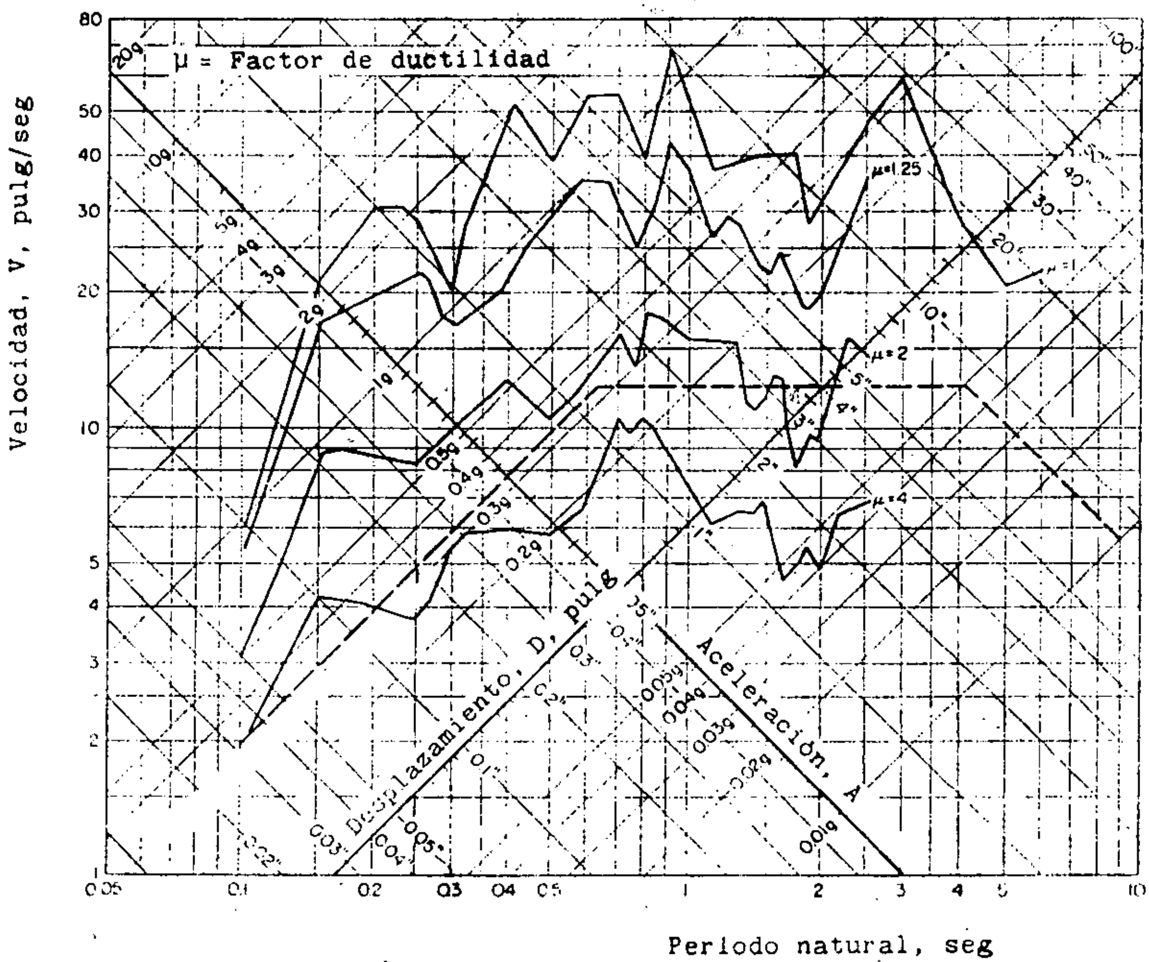
CON $\Delta t = 0.2$ seg Y $\beta= 1/6$, LAS ECUACIONES DEL METODO β DE NEWMARK QUEDAN EN LA FORMA

$$\dot{x}_j(t_{i+1}) = \dot{x}_j(t_i) + 0.1\left[\ddot{x}_j(t_i) + \ddot{x}_j(t_{i+1})\right]$$

$$x_j(t_{i+1}) = x_j(t_i) + 0.1\ \dot{x}_j(t_i) + 0.04\left[\ddot{x}_j(t_i)/3 + \ddot{x}_j(t_{i+1})/6\right]$$



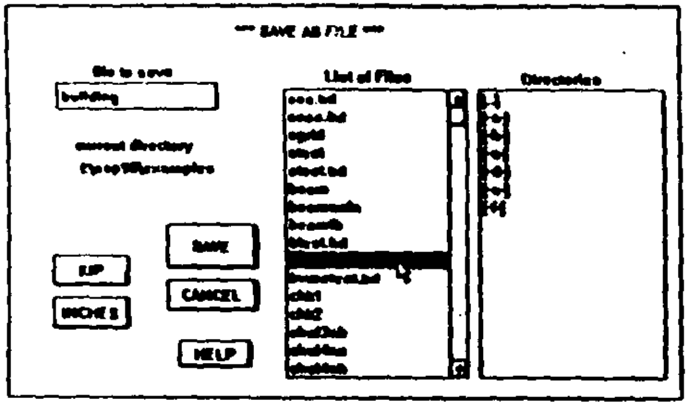
Espectro de respuesta de un sistema elastoplástico con amortiguamiento nulo (parte elástica). Sismo de El Centro, Cal. (1940). Según Blume, Newmark y Corning.



Espectro de respuesta de un sistema elastoplástico con 10% de amortiguamiento (parte elástica). Sismo de El Centro, Cal. (1940). Según Blume, Newmark y Corning.

Dialog Box

A dialog box is a small box that appears as the result of clicking on a menu command such as **Save as** in the **File** menu. In this case the following dialog box appears:



Dialog boxes usually contain a number of boxes which allow you to enter data, perform actions and see lists of files or other items. These boxes are described below.

List Box

In the dialog box example above, there are two list boxes. One is labeled List of Files. It contains a list of the files in the Current Directory, but other list boxes may contain a list of frame element properties, or other items. You can scroll through the list using the scroll bar at the right side of the list box. See Chapter II for how to do this. Note that if there are not enough items to fill the list box, then no scroll bar is shown.

Edit Box

In the SAVE AS FILE dialog box example above, the edit box is the box labeled File to Save. The purpose of edit boxes is to enter data from the keyboard or to display data

from a list box. See Chapter II for a description of entering data. In the example dialog box, clicking on a file in the List of Files list box will put its name in the File to Save edit box.

Push Button

The boxes labeled SAVE, CANCEL, HELP, POUND and INCHES in the dialog box example above, are all push buttons. When seen on screen, these buttons will usually be colored light gray. Push buttons perform actions when clicked on (SAVE, CANCEL, HELP) or allow selection from a set of choices (POUND, INCHES), but you can NOT enter data in a push button. For example, clicking on the SAVE push button saves the current structure in the File to Save which is in the Current Directory.

Radio Button

The Control Panel contains radio buttons under Display and Mouse (see the main SAPIN screen on page I-4). Radio buttons work like the station select buttons on a car radio, that is, when you select one, the others in that group of buttons are turned off. In this case there are two groups of buttons, Display and Mouse, and each group works separately.

Check Box

The Control Panel contains check boxes under Show (see the main SAPIN screen on page I-4). Check boxes are independent of each other, that is, you can check or uncheck any check box without affecting the other check boxes. In this case, a checked box will show the item while an unchecked box will not show it.

II.

USING the MOUSE and KEYBOARD

SAPIN follows the conventions given in the Windows Users Guide for the use of mouse and keyboard. This chapter describes the most commonly used conventions. For more details, refer to the Windows Users Guide. Please note that SAPIN can NOT be run without a mouse, as many of the dialog boxes can not be exited using the keyboard.

Mouse click

Press and quickly release the LEFT button, while the arrow cursor is pointing to the item you are clicking on.

Mouse double-click

Click the LEFT button twice rapidly while the arrow cursor is pointing to the item you want to double-click on. If you have a problem with this, it is probably because you are not clicking fast enough. It does take some practice. If you have a continual problem, the double-click speed can be set slower. The procedure is described in the Windows Users Guide.

Mouse RIGHT click

Press and quickly release the RIGHT button, while the arrow cursor is pointing to the item you are clicking on. RIGHT click is only used to show the properties of joints and elements, described later in this chapter

Mouse Zoom

The mouse is used to reduce the size of the structure displayed in either 2D or 3D views. Zoom must be selected in the Control Panel.

Zoom by drawing a rectangle with the mouse which encloses the area you wish to display. Point to the upper left corner of the rectangle, PRESS and HOLD the left button, drag the rectangle to the desired size and RELEASE the button. The area within the rectangle will then be redrawn so it fills the screen. Note that the display area has a given aspect ratio (height and width) which depends on the monitor you are using. The Zoom rectangle should have about this same aspect ratio. Once Zoomed, you can pan across the structure at the same display scale by using the scroll bars at the right and bottom of the screen.

Mouse Assignment

The mouse is used to assign joints in the 2D view and to assign structural elements to the joint layout in both 2D and 3D views. All assignments REQUIRE that Assign be selected in the Control Panel and that the correct item be selected under Assign in the Control Panel. The assignment procedure is as follows:

Joints - in 2D view only, click on any grid intersection to assign a joint.

Frames - click on the joint at one end and then click on the joint at the other end of the frame.

Shells and Asolids - click on each joint around the perimeter of the element and finish by clicking on the joint you started with. You can go either clockwise or counter-clockwise, but do NOT cross over the element - around the perimeter. This completes the shell assignment, but asolids will ask you if you a c joint, which you y select if

desired. Further, asolids with 5 to 7 joints will require specifying the midside joints. See Asolid Assign in Chapter IV for complete details.

Bridge Lane Loads - click on each joint along the lane in sequence and finish by clicking again on the last joint (not a double click). There must be existing frames between each of the joints.

All other joint assignments (Restraints, Constraints, etc.) are assigned by clicking on individual existing joints.

All other frame assignments (Span Loads, Prestress) are assigned by clicking on the end joints of an existing frame.

See the Control Panel and the **Assign** menu in Chapter IV.

Mouse RIGHT Button Assign Show

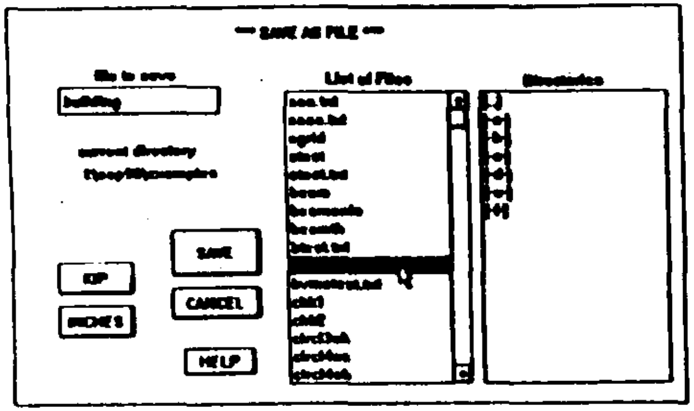
If Assign is selected in the Control Panel, then clicking on any joint with the RIGHT button will display the joint number in the upper right corner of the screen below the menu bar.

You can also show all properties about Joints, Frames, Shells and Asolids by using Right Button Show in the **Display** Menu. Select one of the radio buttons in the Show Select box. Then use the same procedure as for assigning the item but with the RIGHT button, and a dialog box will show the properties of the item. This also requires that Assign be selected in the Control Panel, but it does not matter which item is specifed to be assigned in the Control Panel.

For example, if you have selected Frame in the Right Button Show box, then clicking at each end of an existing frame with the RIGHT button will show the properties of that frame.

Scroll Bars

The following dialog box contains a scroll bar in the list box labeled List of Files:



The small empty box in the scroll bar shows the present position in the list box. Point to the box, press and hold the left button, move the box as desired and then release the button. If the list of items is short, so that they all show, then there is no scroll bar. You can also scroll the contents of the list more slowly by pointing to one of the arrows at the end of the scroll bar and either clicking, or pressing and holding the left button. All scroll bars work in the same way.

Keyboard Data Entry

In the dialog box shown above, you can enter data in the edit box labeled File to Save. This can be done entirely from the keyboard as follows:

Press the Tab key repeatedly and you will see the contents of the edit box highlighted. When highlighted, then type your data on the keyboard. When you type the first character, the edit box contents will be erased and the new data will take its place.

If there is more than one edit box, then the Tab key will cycle through all the boxes in one direction, highlig[illegible]g each box in turn. You can use Shift-Tab to go the other direction.

Another way to enter data uses the mouse and keyboard together. Point the arrow cursor to the edit box and click. The cursor will change to a thin vertical line, but the present edit box contents will remain. If you type characters on the keyboard, they will be INSERTED in the edit box. You can move the line cursor left and right with the arrow keys, the Backspace key deletes the character to the left of the cursor and the Del key deletes the character to the right of the cursor.

Which method to use? It depends. If you have a long entry in the edit box and only want to change one or two characters, then using the mouse is best. Generally, however, using the Tab key is faster as it erases the present contents of the edit box.

III.

TUTORIAL

This tutorial defines the step by step procedure to produce SAP90 models for two simple structures. The tutorial will introduce you to the basic options of SAPIN. For a detailed description of all the options of SAPIN you should refer to Chapter IV of this manual.

The tutorial is composed of two exercises. The first exercise will guide you through the generation of the structural model for a simple truss structure as shown in Figure III-1. The second exercise will guide you through the generation of the structural model for a simple barrel shell roof as shown in Figure III-3.

If you are unfamiliar with the Windows terminology, starting Windows applications or the use of the mouse and the keyboard in association with Windows you should read Chapters I and II of this manual first.

The exercises assume that Microsoft Windows has been installed according to the Windows setup instructions and a path has been established to the directory where the Windows files reside (using the DOS path command.) It is also assumed that the SAPIN disk has been copied onto the harddisk in directory SAP90.

EXERCISE ONE

In order to generate a SAP90 model of the simple truss shown in Figure III-1 you will need to execute the following steps:

a. Start Windows and SAPIN

b. Interactively generate the structural model and loading

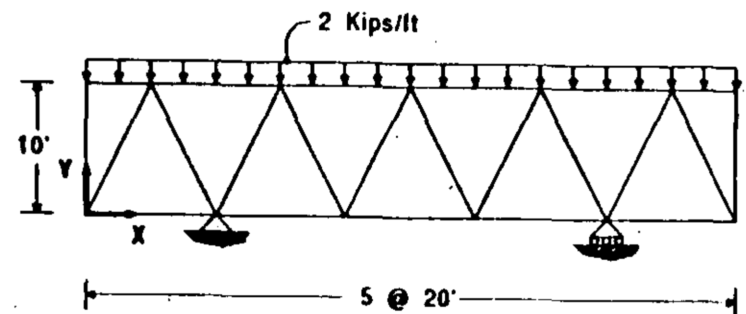
c. Save the SAP90 model

d. Quit SAPIN

e. Quit Windows

The completion of the steps noted above will produce a text file that contains the input data required to execute SAP90 for the analysis of the model generated in SAPIN.

The following five sections (a through e) of this exercise correspond to the steps specified above. Each section defines in detail the procedures required to implement the associated step.



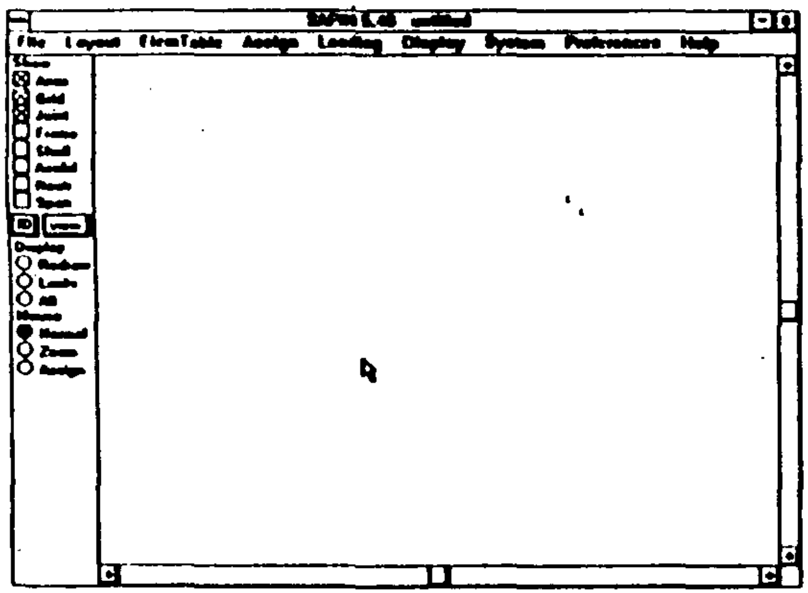
Top and Bottom Chords	2L5 x 5 x 1/2 - 3/8	continuous
All other members	2L3 x 3 x 3/8 - 3/8	pin-ended

Modulus of Elasticity 29000 Ksi

SAMPLE TRUSS
Figure III-1

a. STARTING WINDOWS and SAPIN

Windows and SAPIN can be started in several different ways. A few of the methods are discussed earlier in Chapter I of this manual. Using any one of these methods will activate the SAPIN environment. The screen will initially display the SAPIN window with an hourglass shape as the program initializes the arrays needed for modeling. After the initialization is complete the mouse pointer (arrow) appears indicating the program is ready for modeling and the following screen will show:



This is the SAPIN window as indicated by the title bar at the top. The next bar down (the menu bar) displays the SAPIN main menu items. Each menu item has a corresponding pull down menu that is displayed and accessed by clicking on the menu item (pointing to the item using the mouse and pressing the mouse left button.) Each pull down menu has a list of commands. A particular command from the pull down menu can be selected by clicking on it.

On the left side of the screen is the Control Panel. The Control Panel is a dialog box that is always visible and contains control items that determine what is shown on the screen, when the screen is drawn and how the mouse operates. The Control Panel also allows for quick selection of frequently changed values during assignment of joints and elements.

b. INTERACTIVELY GENERATING THE MODEL

The following are the steps for interactively generating the model:

i. Define a grid system

ii. Define joint locations

iii. Define frame section properties

iv. Locate (Assign) frame elements

v. Assign restraints

vi. Define span loading pattern

vii. Assign span loads

The following subsections (i through vii) of this section correspond to the seven steps specified above. Each subsection defines in detail the procedures required to implement the associated step.

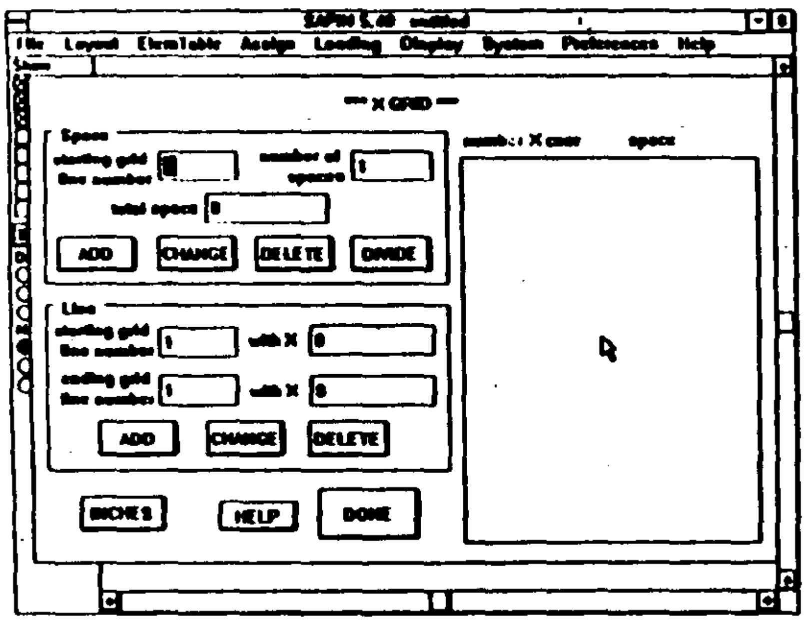
i. Defining a Grid System

The definition of a grid system is not necessary to building a SAPIN model. However, it is highly recommended. If a grid is defined, the joints of the structure can be more easily placed and the model easily visualized when placing the elements.

For 2-D models only two grids X and Y, or Y and Z, or X and Z need to be defined. The third coordinate for all items will be assumed as zero. However, all three grids can be defined.

To define the grids for this problem do the following:

1. Click on **Layout** on the menu bar. The **Layout** pull down menu will appear. Click on **X Grid** in the **Layout** pull down menu. The following **X GRID** dialog box will appear:



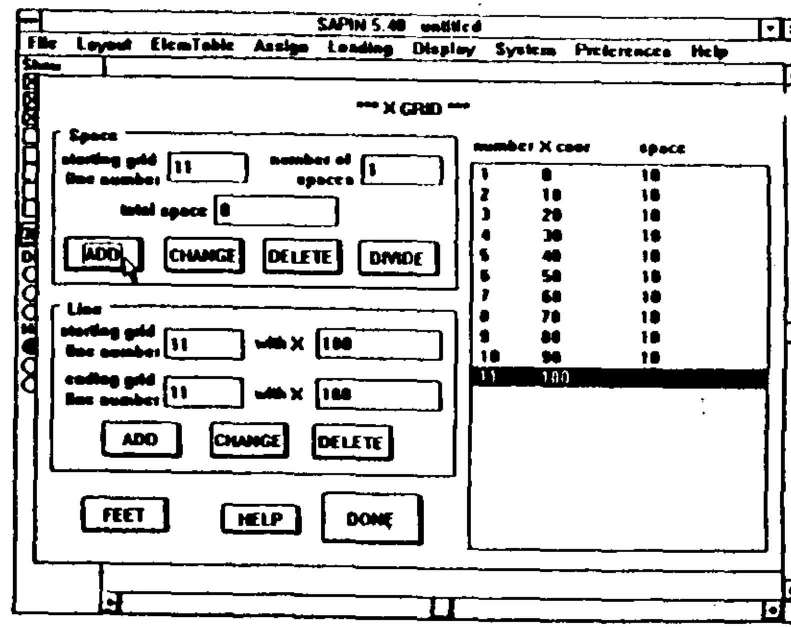
2. The grids can be defined by either adding spaces or grid lines or a combination of both. One method may be easier then the other depending on what proportion of grids are equally spaced. For this problem the grids will be equally spaced and addition of spaces is used.

Type **1** in the edit box labeled **starting grid line number**. Type **10** in the edit box labeled **number of spaces**.

3. Click on the units push button to change it to FEET instead of INCHES.

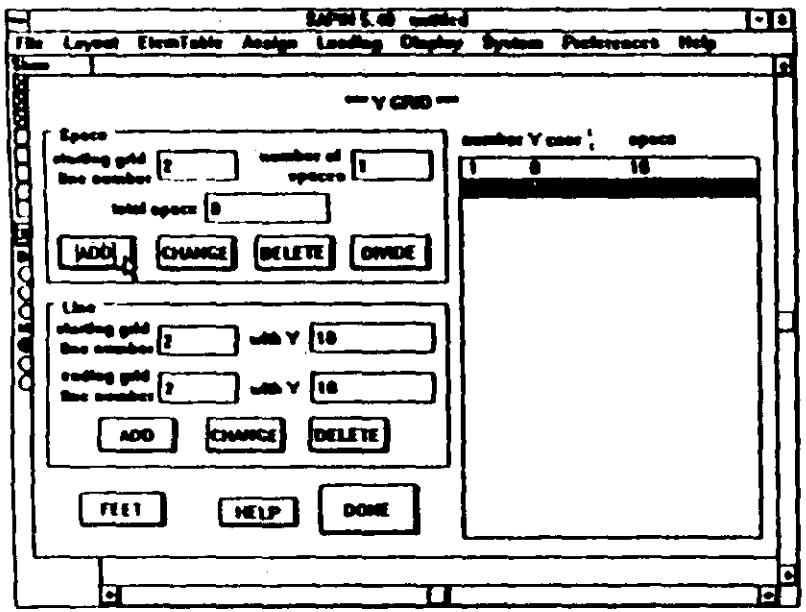
4. Type **100** in the edit box labeled **total space**.

5. Click on ADD. This will add ten equal spaces (grid lines 1 to 11) to the table of X grid lines and the screen will appear as follows:



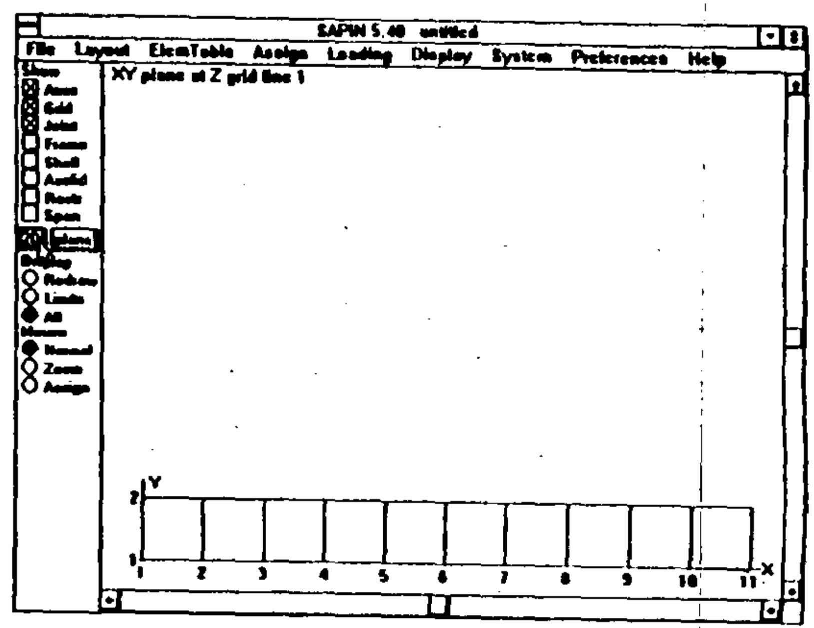
6. Notice that the last grid line in the list box is highlighted and its values are entered in the edit boxes. This is in anticipatition that additional spaces / grids will be added after the last one entered. We do not need to add additional grids for this problem. Click on DONE. The program will close the X GRID dialog box and return with the blank SAPIN window.

7. Repeat the above procedure for the Y grid. Type 1, 1 and 10 in the edit boxes labeled starting grid line number, number of spaces and total space, respectively and then click on ADD. The screen will show as follows:



Click on DONE. The Y GRID dialog box will close and the blank SAPIN window will show.

8. To view the grids click once on the pushbutton labeled 3D in the control panel changing it to 2D. The following screen will show:



Notice the message at the top left of the screen. The program is showing the grid on the XY plane at Z grid line number 1. The program has automatically added a Z grid at a coordinate of zero. Therefore, any joints now assigned to these grid lines will get a Z coordinate of zero. If we want to have a different Z coordinate we should have defined a Z grid earlier or should now go and change the coordinate of the Z grid that the program has defaulted.

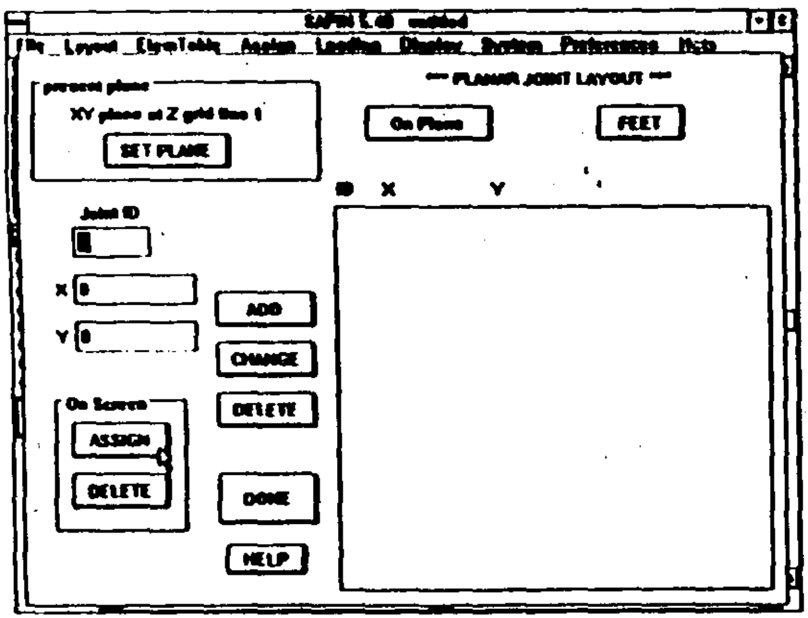
Also note that if we have grids in all three directions we could view any plane at any grid line by choosing the plane and grid in the PLANE SELECT dialog box accessed by clicking on the control panel pushbutton labeled plane.

The definition of the grids for this problem is now complete.

ii. Defining Joint Locations

Several options are available to define joints in SAPIN. We will use the on screen definition accessing it through the layout menu. To define the joints do the following:

1. Click on **Layout** on the menu bar. Click on **Joints** in the **Layout** pull down menu. The following PLANAR JOINT LAYOUT dialog box will appear (Note that the 3D JOINT LAYOUT dialog box will appear instead if the display is set to 3D in the control panel):

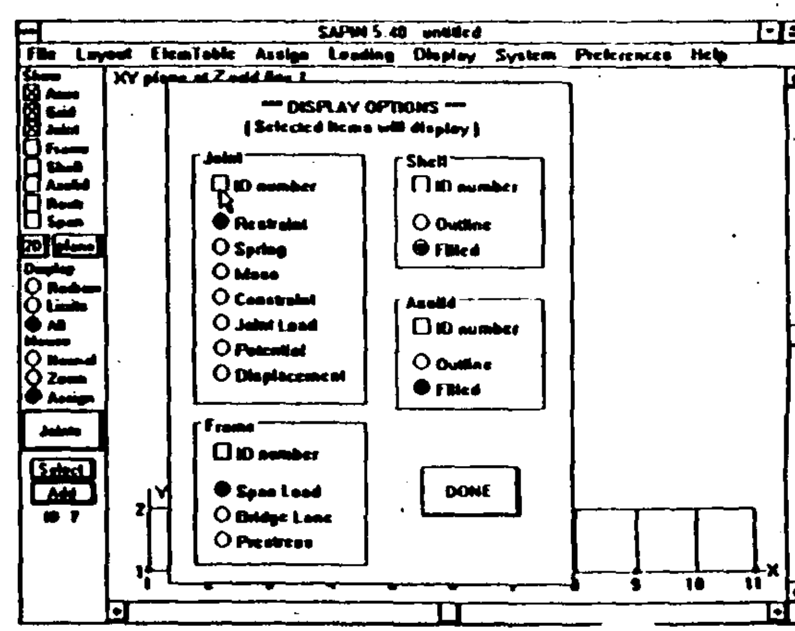


2. Note that the edit box labeled **Joint ID** is initialized to 1. Unless we change it, this will be the ID of the joint we assign. If we assign multiple joints, they will have ID's starting from this number and incremented by 1. Since we wa[illegible] assign joints starting with an ID of 1, we will leave th[illegible]ault as is. Click on ASSIGN in the ON SCREEN box. The program will now show the [illegible] id [illegible] is [illegible] y to add

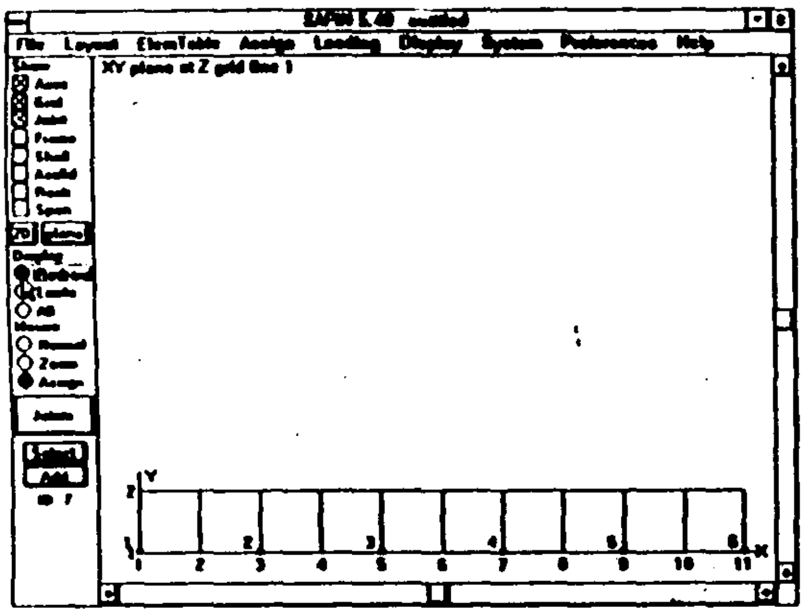
joints. Note that Assign is now on in the Control Panel and that other items related to the assignment being made are now showing.

3. Click on the grid intersections (1,1), (3,1), (5,1), (7,1), (9,1) and (11,1) along Y grid 1 to assign joints to these intersections. The program will display a small square to show that a joint now exists at these locations. The ID number on the Control Panel will also keep on incrementing, showing the ID number of the next joint to be assigned.

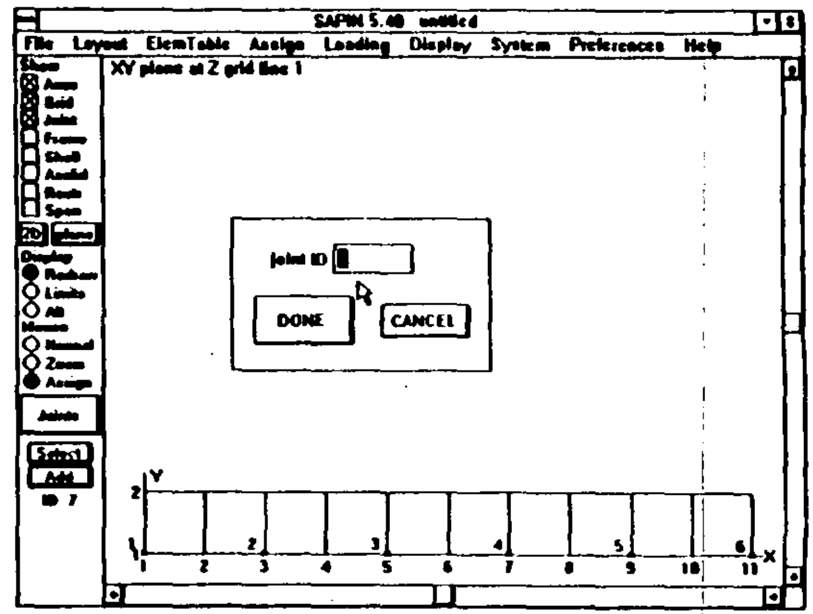
4. The normal default for the program is not to show joint numbers on the screen. Since we are adding joints we may wish to view the joint ID's being assigned. To show the joint numbers click on **Display** on the menu bar and then click on **Display Options** in the **Display** pull down menu. The following screen will show.



5. Click on the check box labeled ID number in the Joint section once to check it. Click on DONE. The DISPALY OPTIONS dialog box will close and the SAPIN window with the grid and joints will show. The joint numbers are still not visible because the program does not automatically redraw the display. Click on Redraw in the Control Panel. The following screen will show:

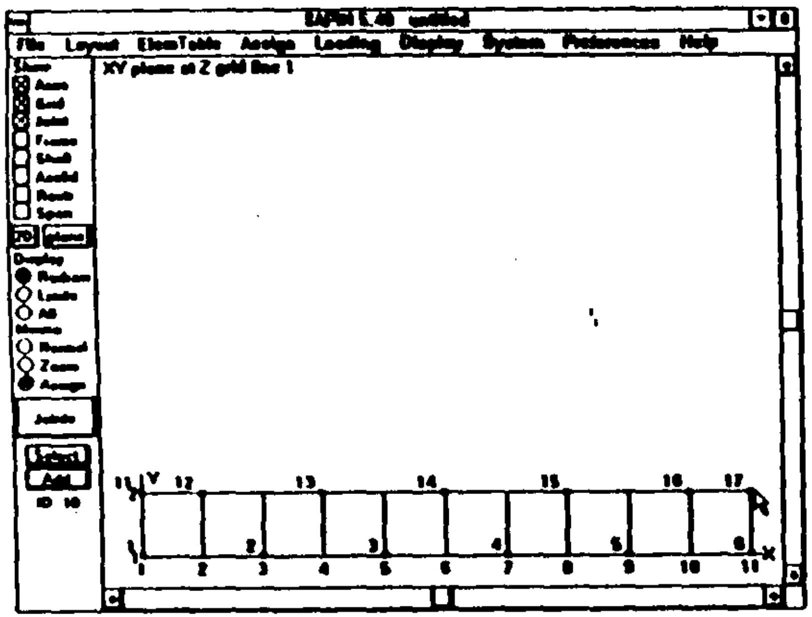


6. We will use a different numbering scheme for the top row of joints. To do this click on Select in the Control Panel. The following screen will show:



7. Type in 11 in the edit box labeled Joint ID. Click on DONE. The ID number showing in the Control Panel will change to 11.

8. Click on grid intersections (1,2), (2,2), (4,2), (6,2), (8,2), (10,2) and (11,2) to complete the joint assignments. The following screen will show:

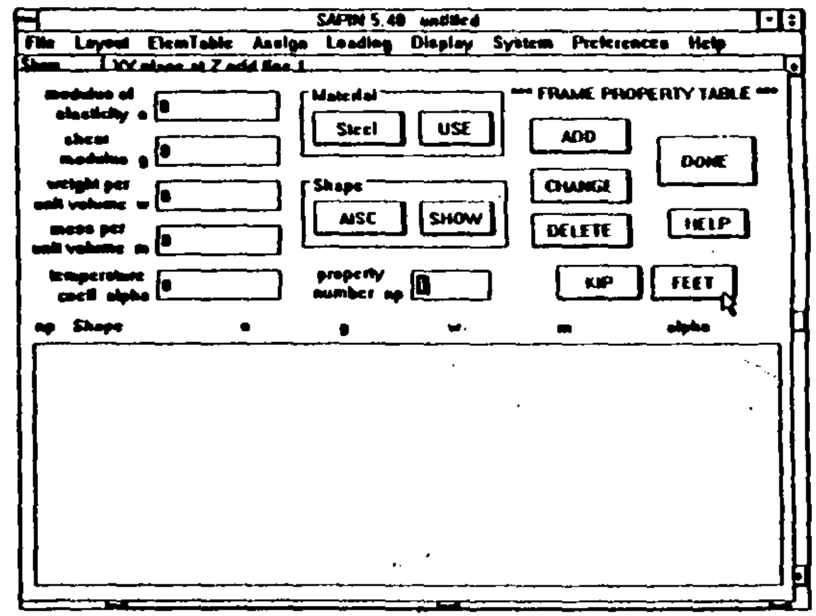


The location of the joints is now complete.

iii. Defining Frame Section Properties

To define frame element section and material properties do the following:

1. Click on **ElemTable** on the menu bar. Click on **Frame** in the **ElemTable** pull down menu. The following FRAME PROPERTY TABLE dialog box will appear:



2. Click on the length unit push button a few times to change it to INCHES.

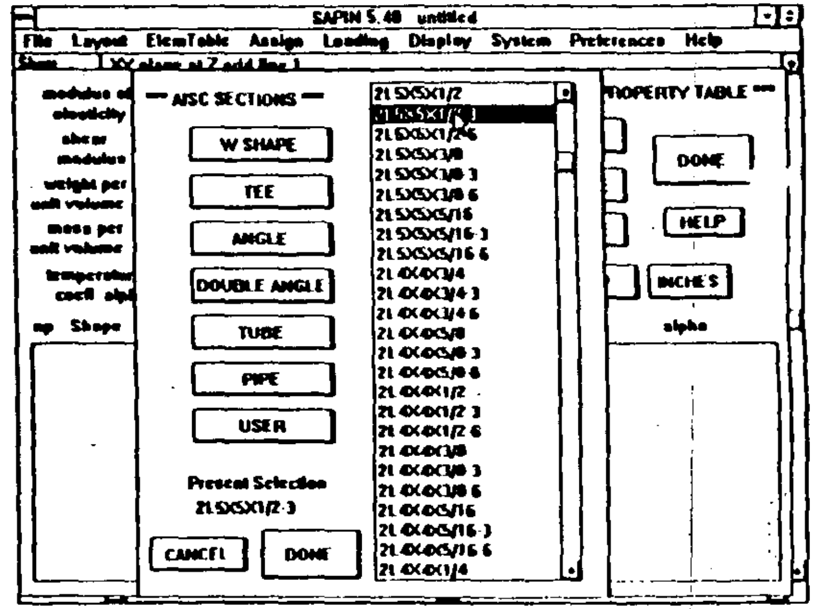
3. Material properties can be input in SAPIN by directly typing them in or by using built in properties. For this problem we will use the built in properties. Noting that under Material Steel is showing, click on USE. The built in properties for steel will be brought into the edit boxes.

4. Section properties can also be input in SAPIN several different ways. For this example we will simply pick out the shape from the AISC database. Noting that under Shape AISC is showing, click on SHOW. The AISC SECTIONS dialog box will appear.

5. Click on the DOUBLE ANGLE push button and the program will list the available double angle shapes.

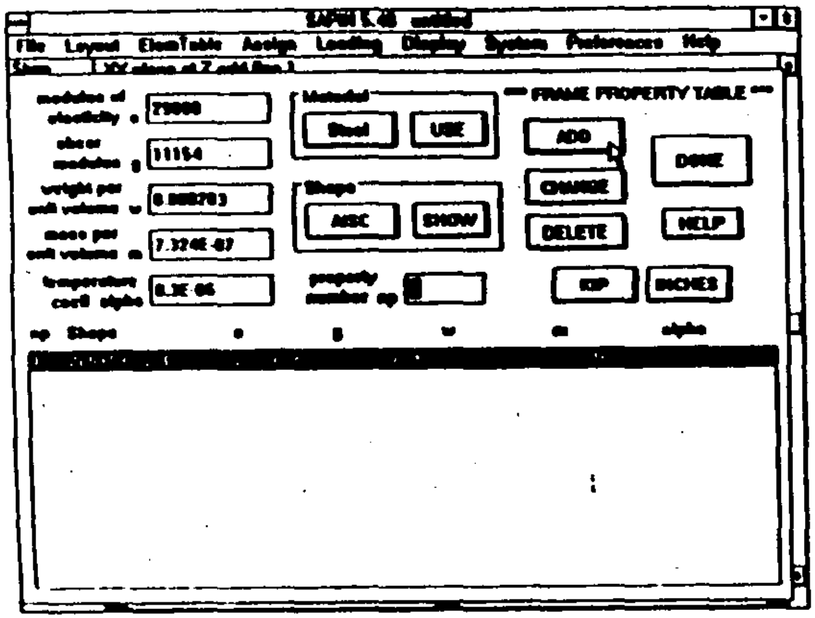
6. Use the scroll bar showing on the right of the list of labels to move up or down the list until 2L5 x 5 x 1/2 - 3 is in view. The scroll bar can be used by either clicking on the arrows at the ends to move the list a little at a time, or by pointing to the box within the scroll bar and pressing the mouse left button, moving the pointer with the mouse button down (the box will move with it) and releasing the mouse button where we want to locate the box. The location of the box along the vertical scroll bar shows the location of the list in view compared to the full list.

7. Click on 2L5x5x1/2-3 to select it. The screen will show as follows:

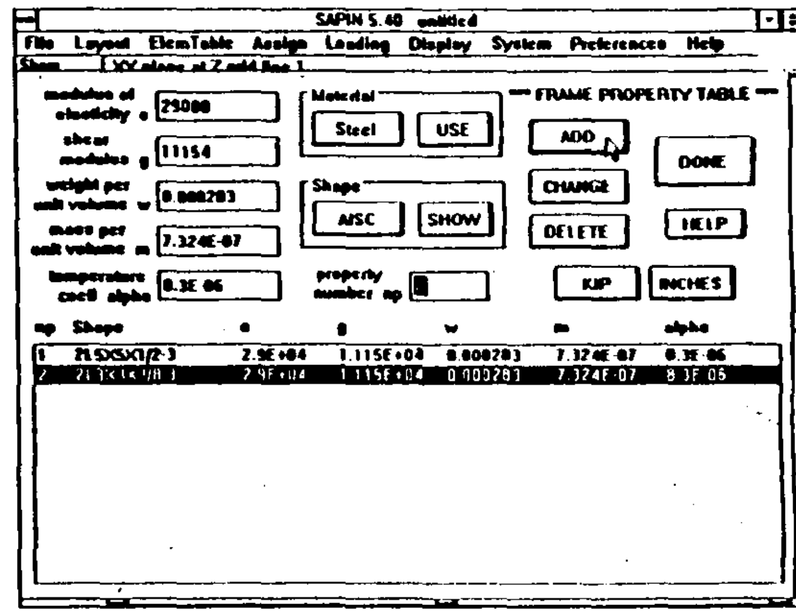


8. Click on DONE. The program will close the AISC SECTIONS dialog box and will return to the FRAME PROPERTY TABLE dialog box.

9. Click on ADD. The program will enter the selected section in the frame element table and the screen will appear as follows:



10. Repeat the above procedure for the second section. Type 2 for the second property in the property number np edit box; click on SHOW to bring back the AISC SECTIONS dialog box; select 2L3x3x3/8-3 from the list of double angles; close the AISC SECTIONS dialog box; and click on ADD to add the second section to the list of frame sections. The screen will appear as follows:

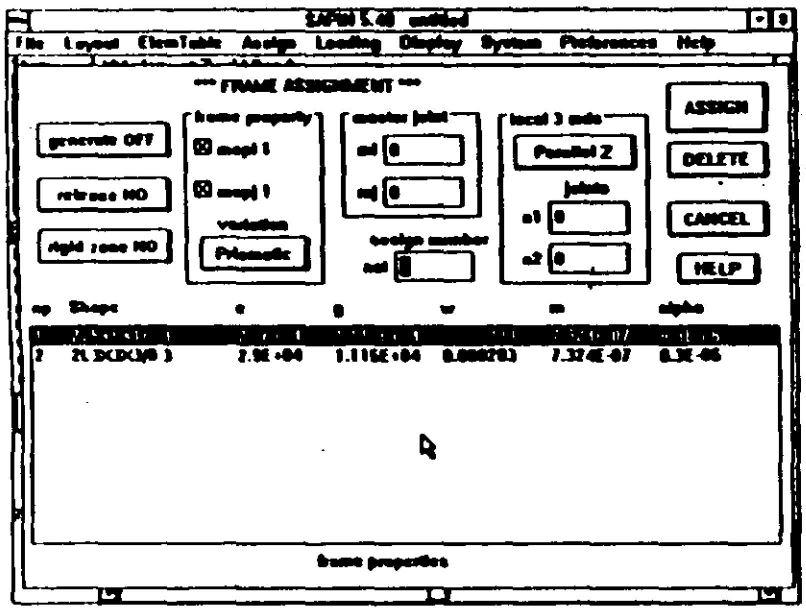


11. Click on DONE. The program will close the FRAME PROPERTY TABLE dialog box and return the earlier SAPIN window.

iv. Locating Frame Elements

To locate (assign) frame elements between the joints do the following:

1. Click on **Assign** on the menu bar. Click on **Frame** in the Assign pull down menu. The following FRAME ASSIGNMENT dialog box will appear:

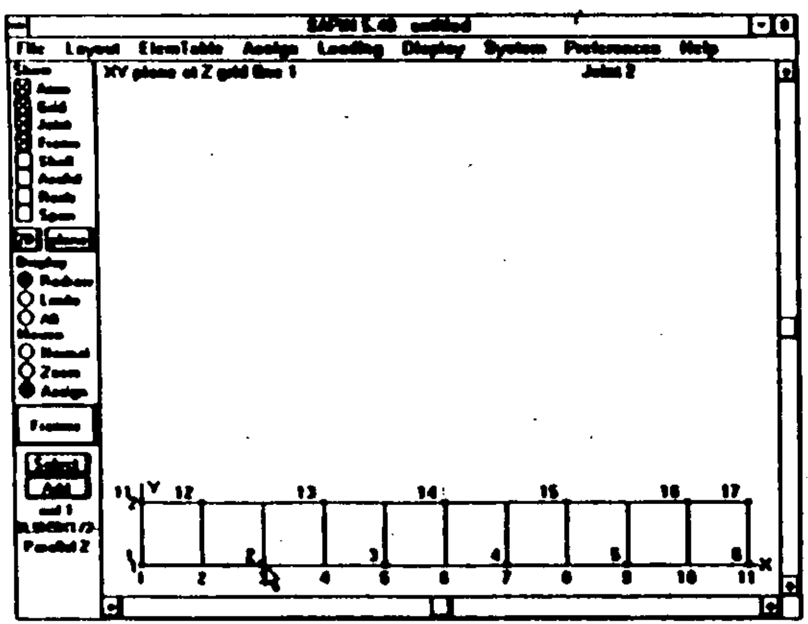


2. We will first locate the top and bottom chords which are continuous. Click on the property line in the list box which we want to assign to the chords. In this case it is property number 1 (2L5x5x1/2-3). This would then show **mspi** and **mspj** the section property numbers at start and end of the member as equal to this number. This is because the variation is shown as Prismatic. If the member was non-prismatic we should first select its variation. The program would then allow different properties to be picked for the member start and end.

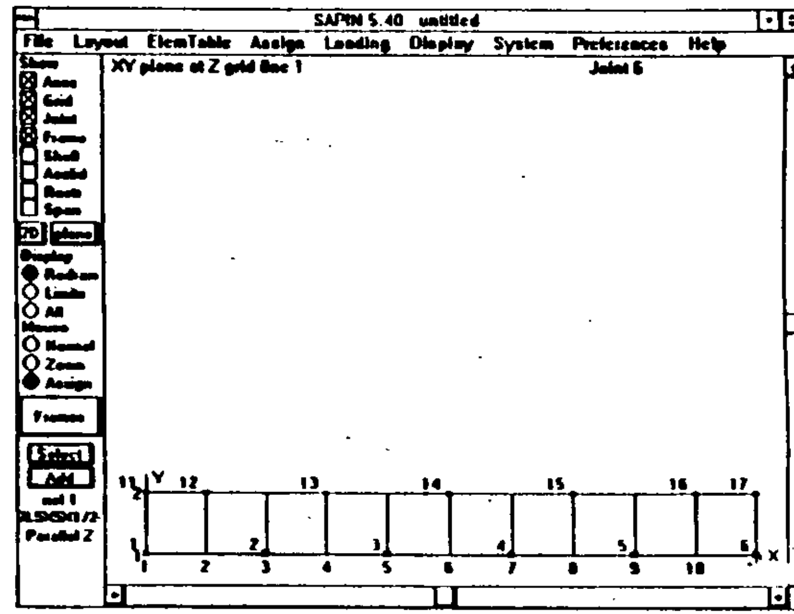
3. Let us now check some of the defaults for applicability. We see that the **generate** button shows it is OFF. This is the correct setting for this example as we would assign members singly. If this was not the case clicking on generation would bring up another dialog box to set up the generation parameters. The **release** button shows NO. This is correct for the chord members we are going to assign as they are continuous. The **rigid zone** button shows NO. This is correct for this example as we do not want to model rigid end offset (to model connection sizes). The **master joints mi** and **mj** are set to zero. This is correct as we do not want to model any rigid diaphragm. The start **assign number nel** is set to 1. This is correct as we will start assigning element ID numbers from 1. The **local 3 axis** is shown as Parallel Z. This is needed to define the local coordinate system for the frame elements for the program to properly assign bending properties and any span loads defined in the local system. Several methods are available in the SAP90 (and SAPIN) program to do that. We will assign the chord members going in the positive X direction. This becomes the local 1 axis. By declaring the local 3 axis to be parallel to the Z axis we would automatically make the local 2 axis parallel to the Y axis. So all of the defaults are what we want.

4. Click on ASSIGN. The program will now close the FRAME ASSIGNMENT dialog box and will show a screen with the joints showing.

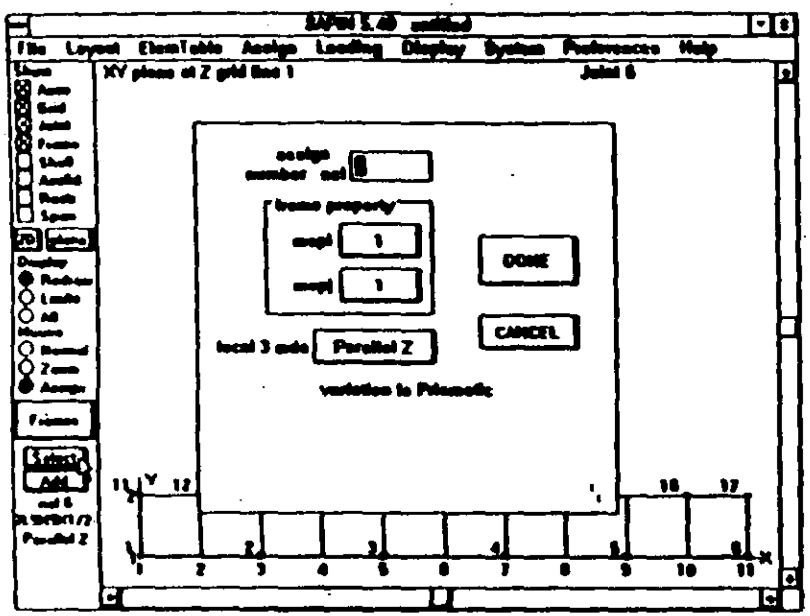
5. To assign a frame element we need to click on the two joints to which the element is connected. To assign the first element of the bottom chord, we need to click on joints 1 and 2. The program draws a line between them indicating a member has been assigned. Notice that at the top of the screen some messages are given. At the right the joint number most recently clicked is shown. In the center a message tell you how many frame element joints have been clicked on for this element. Also notice that the element ID number nel in the control panel got incremented from 1 to 2 in anticipation of the next element to be assigned. The screen now shows as follows:



6. Clicking on joints 2, 3; 3, 4; 4, 5; and 5, 6 will complete the assignment of elements for the bottom chord. The screen will show as follows:



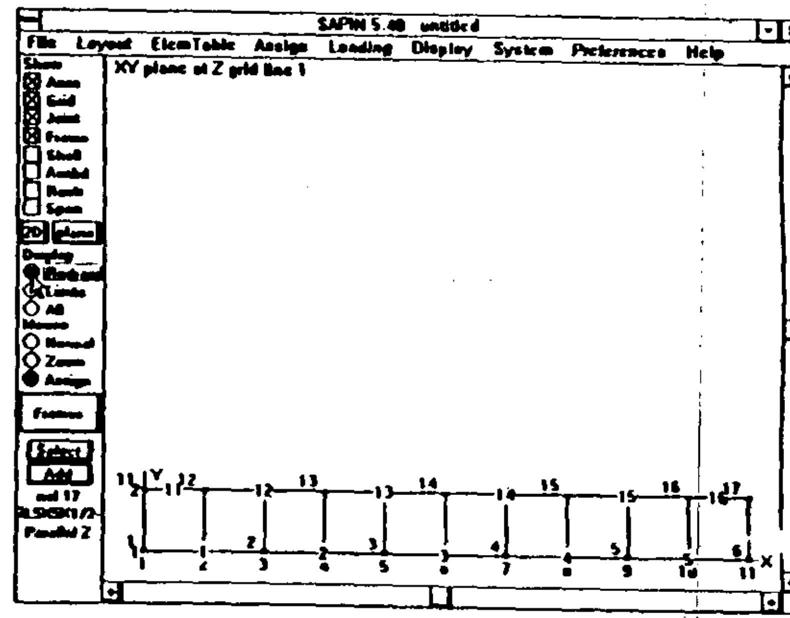
7. To change the element ID to be assigned to the top chord instead of simple incrementing by 1, click on **Select** in the Control Panel. A selection box will show as follows:



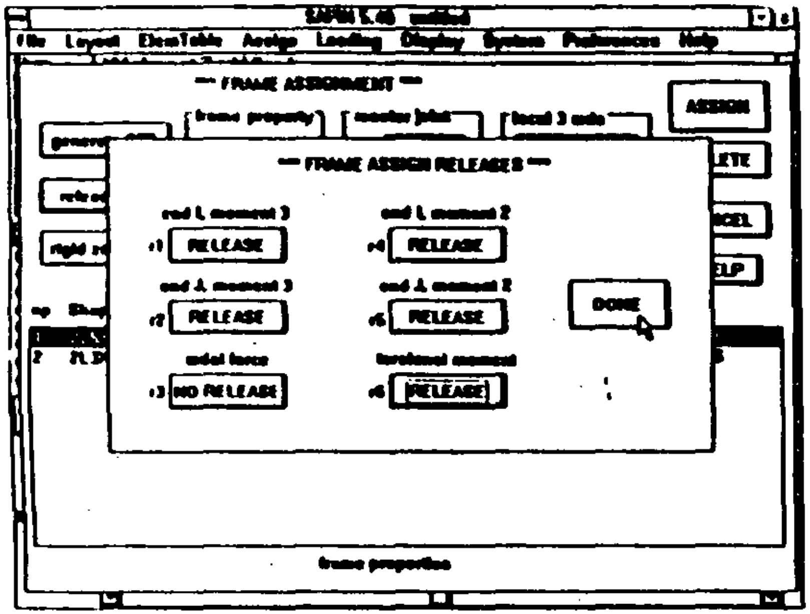
8. This allows for easily changing the assign number (ID number) of the element and its section properties. We only want to change the assign number. Type in **11** in the edit box labeled **assign number nel**, to start the top chord element ID's from 11. Click on DONE to close the dialog box.

9. We are now ready to assign the top chord elements. Click on Joints 11, 12; 12, 13; 13, 14; 14, 15; 15, 16; and 16, 17 to assign the top chord elements.

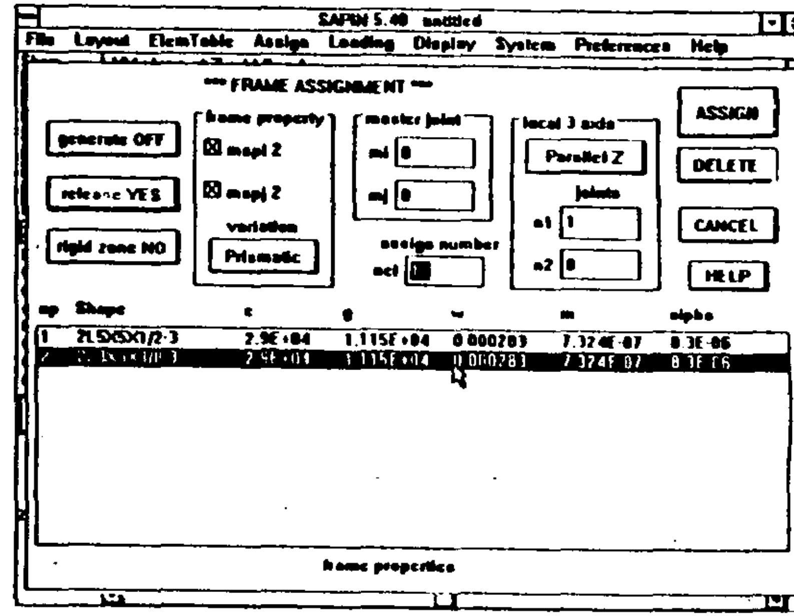
10. To view the frame element ID numbers, click on **Dispaly Options** in the **Display** pull down menu. The DISPLAY OPTIONS dialog box will show. Click **ID number** under frame. Click on DONE to close the dialog box. Now click on **Redraw** in the Control panel. The following screen will show:



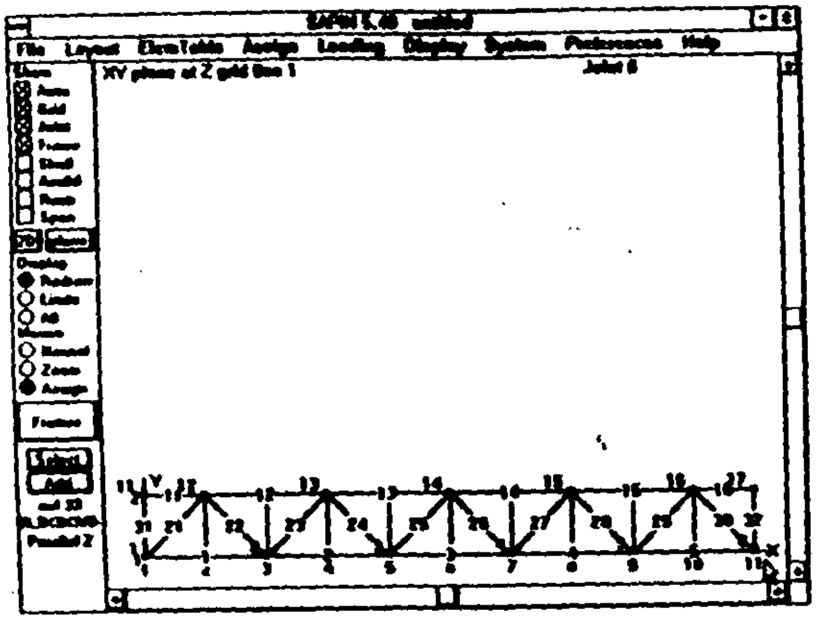
11. The above procedure needs to be repeated for locating the other elements. Click on **Frame** in the **Assign** pull down menu; click on the **Release** push button in the FRAME ASSIGNMENT dialog box; and push on the button for the releases in the FRAME ASSIGN RELEASES dialog box so that all forces are released from the element except the axial force as shown in the screen below:



12. Close the FRAME ASSIGN RELEASES dialog box by clicking on DONE; and select the second section property (2L3x3x3/8-3) by clicking on it in the FRAME ASSIGNMENT dialog box. The screen will appear as follows:



13. We will start the ID numbers for the diagonals from 21, so type 21 in the assign number nel edit box. Click on ASSIGN. The FRAME ASSIGNMENT dialog box will close and the current model of the structure will show. Click on Joints 1, 12; 12, 2; 2, 13; 13, 3; 3, 14; 14, 4; 4, 15; 15, 5; 5, 16; 16, 6; 11, 1; and 17, 6 to assign all the secondary elements. The screen will show as follows:



The location of the frame elements is now complete.

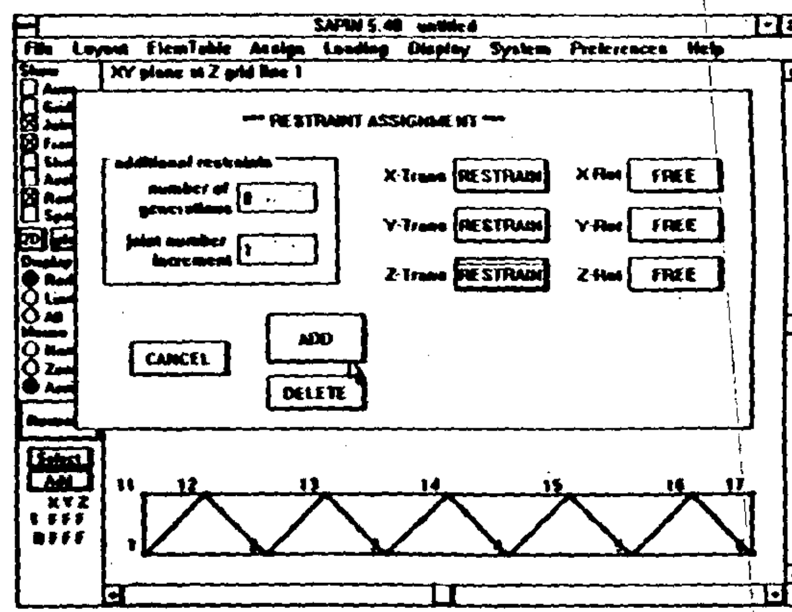
v. Assigning Restraints

The SAP90 program assumes all joints to be free to move three dimensional space unless restraints (support condition are specified. We need to specify the hinge and roller suppon for the two joints which have them and also specify support out of plane at all joints to reduce the model to a two-dimen sional truss.

To specify restraints do the following:

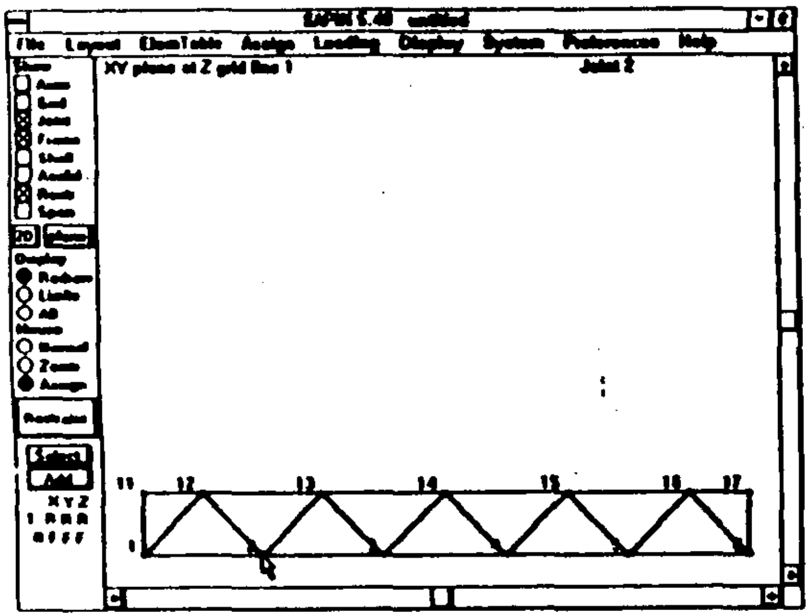
1. Click on Assign on the menu bar. Click on Restraint in the Assign pull down menu. The RESTRAINT ASSIGN-MENT dialog box will appear.

2. Click on the translational push buttons to change them to RESTRAIN. This models the hinge support. The screen will appears as follows:

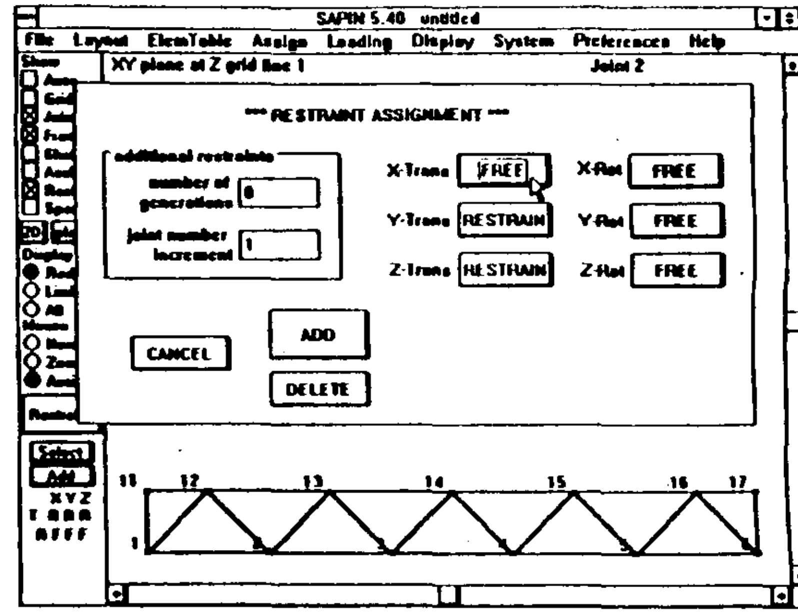


3. Click on ADD. The RESTRAINT ASSIGNMENT dialog box will close and the current model will show on the screen.

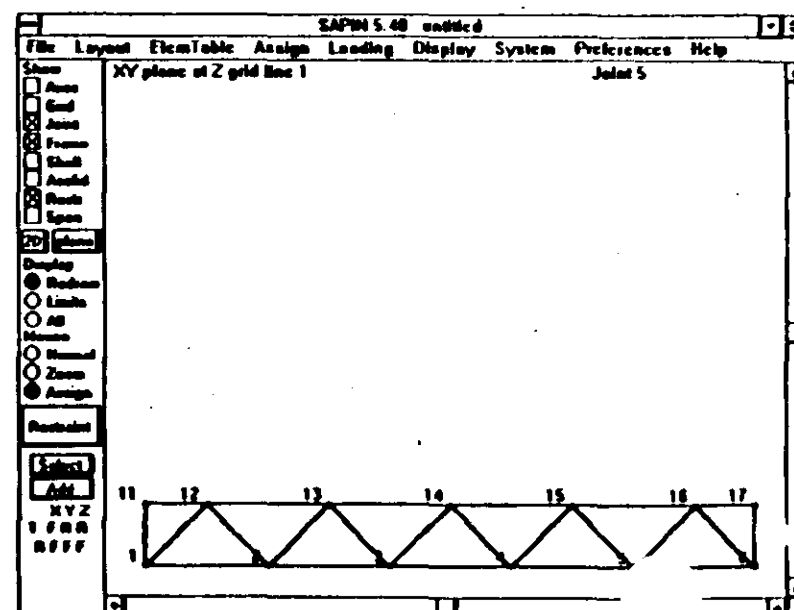
4. Click on Joint 2 to assign the restraints to it. The color of this joint will change. The screen will appear as follows:



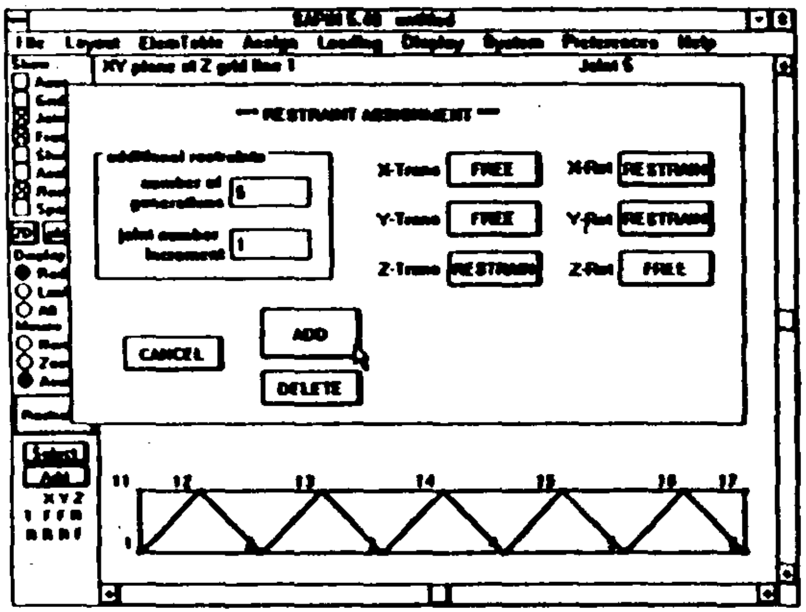
5. Repeat the above procedure for the roller support, except the RESTRAINT ASSIGNMENT dialog box should have the following settings:



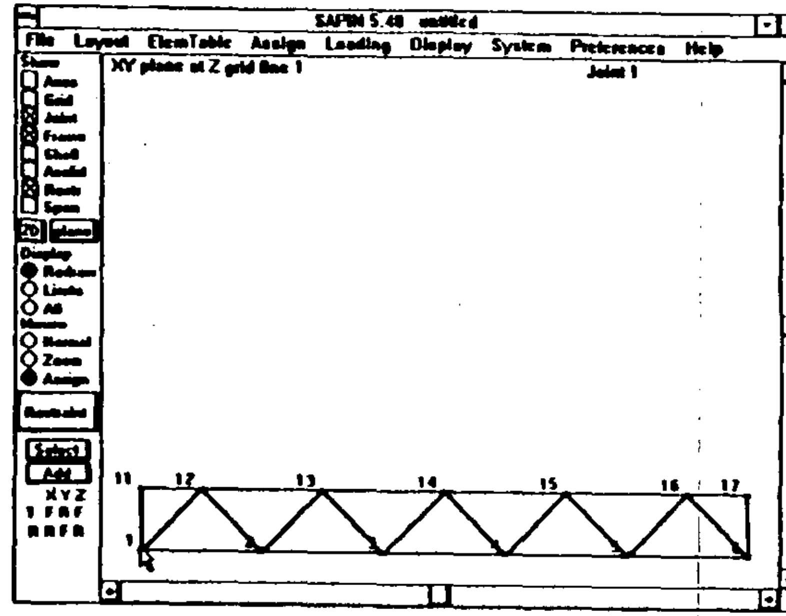
and the roller support should be assigned to Joint 5 as shown in the following screen:



6. To restrain the truss from moving out of plane we can assign out of plane restraints to each joint by repeating the above procedure. (Multiple assignments can be made with one setting by simply clicking on all the different joints that have the same restraints.) However, for this example we will use the automatic generation option of the program. In the RESTRAINT ASSIGNMENT dialog box type in 5 in the number of generations edit box for 5 additional restraints to be generated. Set the restraints as shown below to restrict out of plane motion:



7. Assign this restraint to Joint 1. The program will automatically assign 5 additional restraints to Joints 2,3,4,5 and 6. The screen will now look as follows:



8. Repeat the above steps 6 and 7 for the joints in the top chord. Use 6 additional generations instead on 5 and click on Joint 11 instead of 1.

The assighment of restraints is now complete. Click on Redraw in the Control Panel to obtain a clean view of the model.

vi. Defining Span Loading Patterns

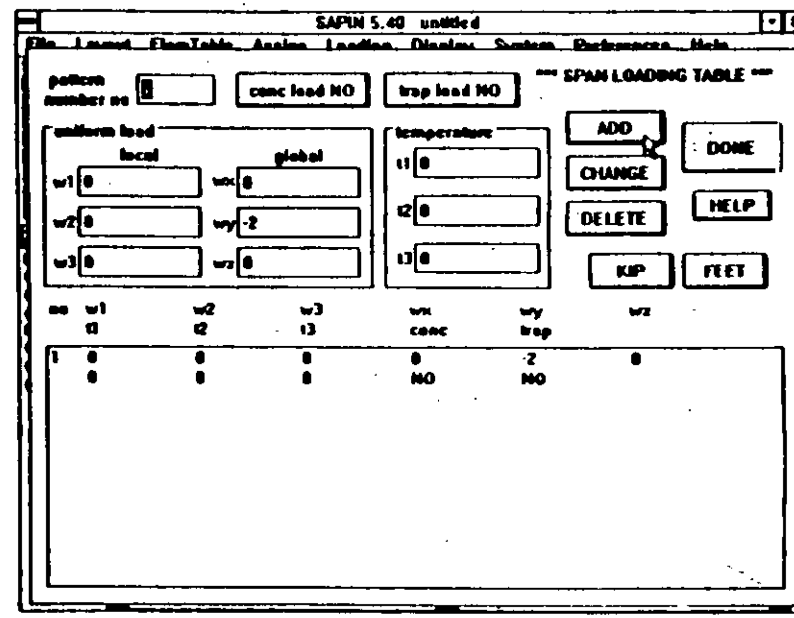
To assign loads onto a frame member in SAP90 (and SAPIN) we need to define the loading patterns first. To define the load pattern for this example do the following:

1. Click on **ElemTable** on the menu bar. Click on **Span Load** in the **ElemTable** pull down menu. The SPAN LOADING TABLE dialog box will appear.

2. Click on the length unit push button to change it to FEET.

3. Type in -2 in the **wy** edit box for the 2 kips/ft load acting downwards (- Y direction) on the upper chord.

4. Click on ADD. The program will add this loading pattern to the list of loading patterns and the screen will appear as follows:

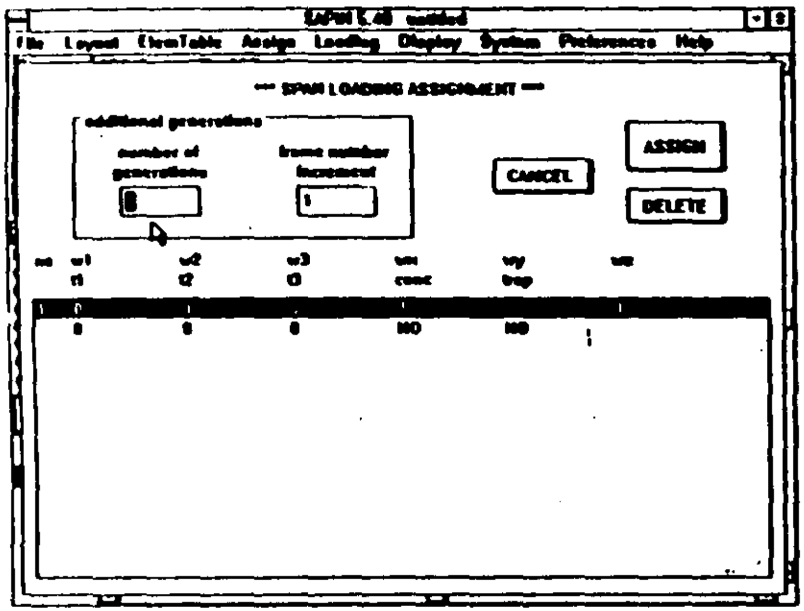


5. Click on DONE. This will close the SPAN LOADING TABLE dialog box and the earlier SAPIN window will appear.

vii. Assigning Span Loads

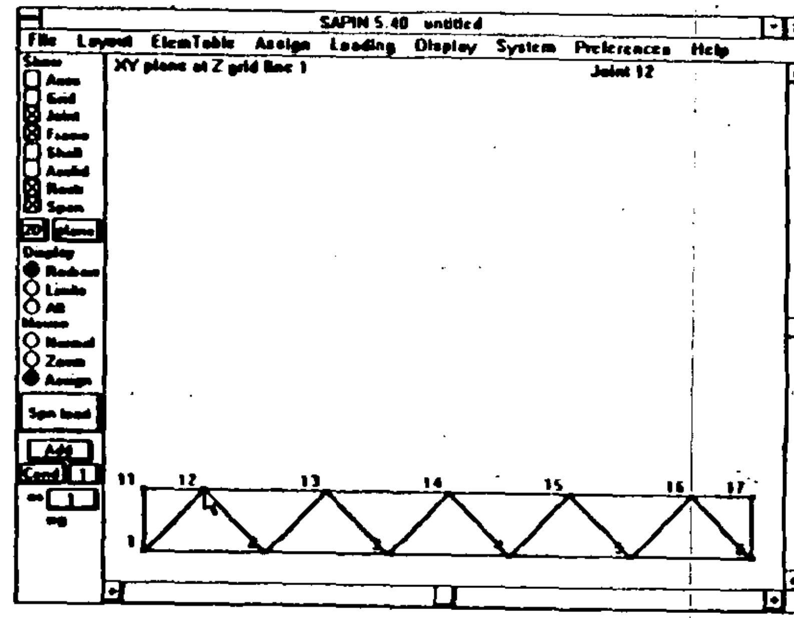
To assign the span loading patterns to the frame elements do the following:

1. Click on **Loading** on the menu bar. Click on **Span Load** in the **Loading** pull down menu. The following SPAN LOADING ASSIGNMENT dialog box will appear:



2. Click on the loading pattern to be assigned to select it and then click on ASSIGN. The program will then show the current model of the structure and will be ready for span load assignments to the frame elements.

3. Assign span loads on frame elements in exactly the same manner as locating frame elements by clicking on the two joints of the frame, element by element. The program will change the color of the frame element on which span loads have been assigned. When all elements of the top chord of our example truss have been assigned span loads the screen will appear as follows:

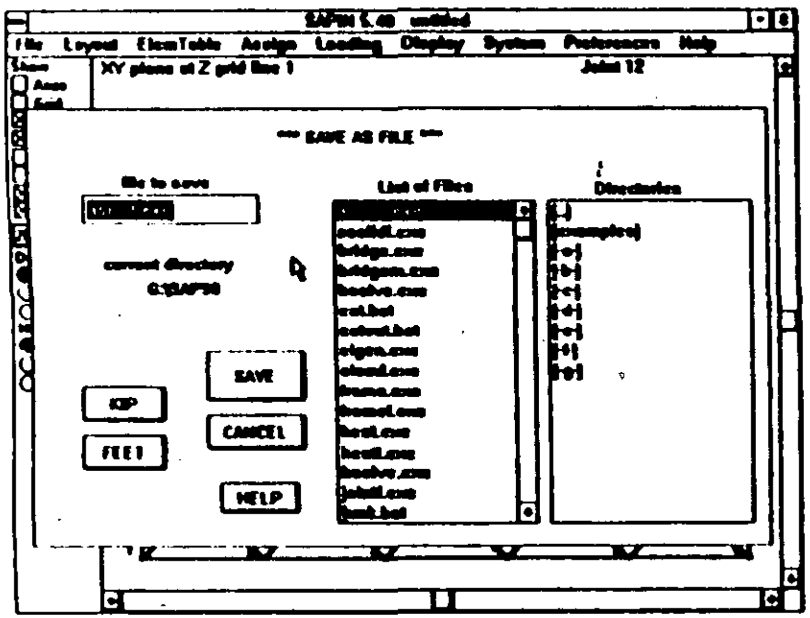


It is important to note here that all loads were automatically assigned to the first load condition which is the default. If loads need to be assigned in other load conditions, the load condition needs to be selected first in the LOAD CONDITION dialog box which is opened by clicking on **Load Condition** in the **Loading** pull down menu, before assigning the span loads.

c. SAVING THE MODEL

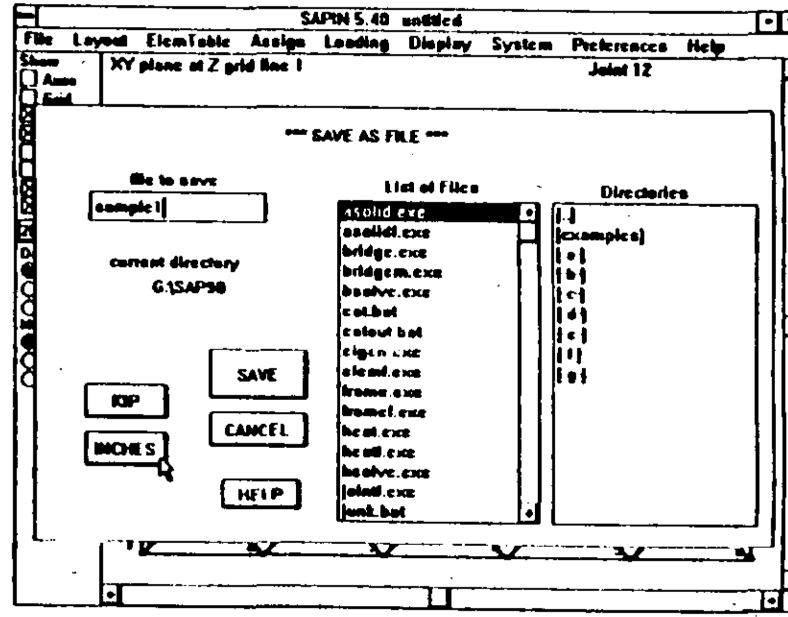
The modeling of the truss is now complete. We can now save the model. It should be pointed out here that intermediate models can also be saved and later brought back into SAPIN for editing and completion. For any significant size problem one should save quite often while working on a model. To save the model do the following:

1. Click on **File** on the menu bar. Click on **Save as** in the File pull down menu. The SAVE AS FILE dialog box will show as follows:



2. Type **sample1** in the FILE to SAVE edit box.

3. Click on the length units push button a few times to change it to INCHES. The screen will now show as follows:



4. Noting that the units, filename and directory are as we wish, click on SAVE. The program will now save the model in an ASCII file called SAMPLE1. The file is in a format directly readable by SAP90 for analysis and by SAPIN for further editing. The program will then close the SAVE AS FILE dialog box and return the earlier SAPIN window.

d. QUITTING SAPIN

To quit SAPIN double click on the small box on the left upper corner (called the Control-menu box) of the SAPIN window. Refer to the Windows Users Guide for other methods of quitting applications. The program will close itself and return you to the Desktop, Program Manager or File Manager based on how the program was started.

e. QUITTING WINDOWS

To quit Windows you may first need to close all active application windows. Double click on the small box on the left upper corner (Control-menu box) of these applications to close them. Once the applications are closed, double clicking on the Control-menu box of the Program Manager quits Windows. If the Program Manager is iconized, clicking on its icon will open its Control-menu. In this case click on Close in the Control-menu to quit Windows. Windows will confirm that you want to quit it. After you confirm by clicking on OK, the Windows program will return you to DOS. Refer to the Windows Users Guide for other methods of quitting Windows.

There is now a file named SAMPLE1 in the SAP90 directory. If the file is viewed on the screen or printed it will appear as shown in Figure III-2.


```
C This is file SAMPLE1 written by SAPIN on Tue Mar 10 12:17:14 1992
C Units are KIP INCHES
SYSTEM
 R=0 L=1 C=0 V=0 T=0.0001 P=0 W=0 Z=0
GRID
 XN=11 YN=2 ZN=1
 0 120 240 360 480 600 720 840
 960 1080 1200
 0 120
 0
JOINTS
 1        X=0 Y=0 Z=0
 2        X=240 Y=0 Z=0
 3        X=480 Y=0 Z=0
 4        X=720 Y=0 Z=0
 5        X=960 Y=0 Z=0
 6        X=1200 Y=0 Z=0
 11       X=0 Y=120 Z=0
 12       X=120 Y=120 Z=0
 13       X=360 Y=120 Z=0
 14       X=600 Y=120 Z=0
 15       X=840 Y=120 Z=0
 16       X=1080 Y=120 Z=0
 17       X=1200 Y=120 Z=0

FRAME
 NM=2 NL=1 NSEC=0
 1   SH=2L5X5X1/2-3       E=29000 G=11154 W=0.0076885 M=6.9578E-06 TC=6.5E-06
 2   SH=2L3X3X3/8-3       E=29000 G=11154 W=0.0011963 M=3.0907E-06 TC=6.5E-06
 1   WL=0,0,0 WG=0,-0.16667,0 T=0,0,0
 1      1    2     M=1,1,1 LP=1,0
 2      2    3     M=1,1,1 LP=1,0
 3      3    4     M=1,1,1 LP=1,0
 4      4    5     M=1,1,1 LP=1,0
 5      5    6     M=1,1,1 LP=1,0
 11     11   12    M=1,1,1 LP=1,0 NSL=1
 12     12   13    M=1,1,1 LP=1,0 NSL=1
 13     13   14    M=1,1,1 LP=1,0 NSL=1
 14     14   15    M=1,1,1 LP=1,0 NSL=1
 15     15   16    M=1,1,1 LP=1,0 NSL=1
 16     16   17    M=1,1,1 LP=1,0 NSL=1
 21     1    12    M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 22     12   2     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 23     2    13    M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 24     13   3     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 25     3    14    M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 26     14   4     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 27     4    15    M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 28     15   5     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 29     5    16    M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 30     16   6     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 31     11   1     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1
 32     17   6     M=2,2,1 LP=1,0 LR=1,1,0,1,1,1

RESTRAINTS
 2    2    1    R=1,1,1,0,0,0
 5    5    1    R=0,1,1,0,0,0
 1    6    1    R=0,0,1,1,1,0
 11   17   1    R=0,0,1,1,1,0
```

File SAMPLE1
Figure III-2

EXERCISE TWO

This exercise introduces the user to some of the joint and element generation options of SAP90 (and SAPIN.) The simple barrel shell roof to be modeled is shown in Figure III-3. The steps required to start Windows and SAPIN and later to quit SAPIN and Windows are identical to Exercise One. After SAPIN has been started the following steps are necessary to interactively generate and save the model:

i. Define starting joints

ii. Generate remaining joints

iii. Define shell element material properties

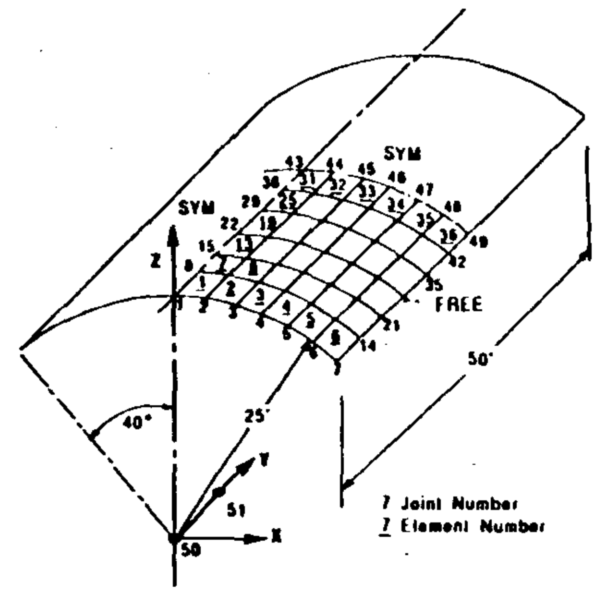
iv. Locate (assign) shell elements

v. Assign restraints

vi. Assign loading

vii. Save the model

The following subsections (i through vii) of this section correspond to the seven steps specified above. Each subsection defines in detail the procedures required to implement the associated step.



THICKNESS	= 3 in
MODULUS OF ELASTICITY	= 4.32 $\times 10^{8}$ psf
POISSON'S RATIO	= 0.0
GRAVITY LOAD (UNIFORM ON SURFACE AREA)	= 90 psf

BOUNDARY CONDITION: SIMPLY SUPPORTED ON CURVED EDGES

EXAMPLE SHELL
Figure III-3

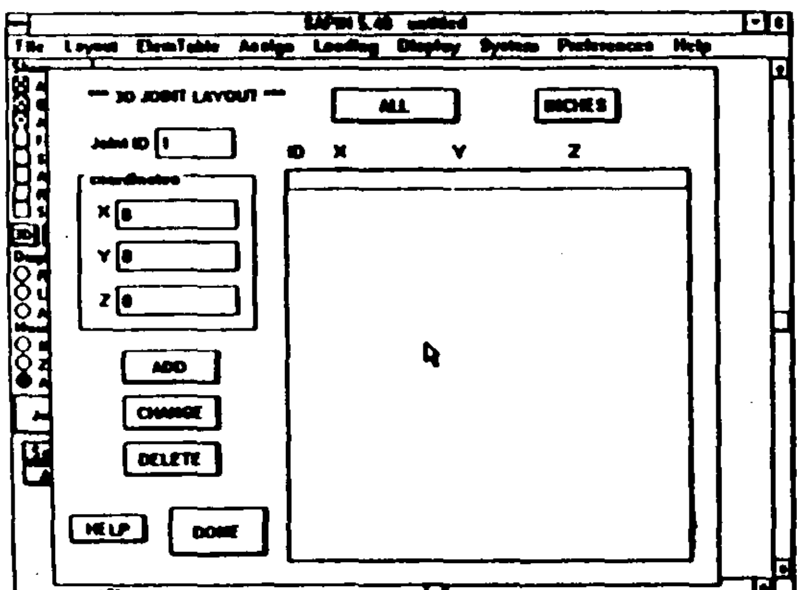
i. Defining Starting Joints

The automatic joint generation features of SAP90 (and SAPIN) require a few starting joints. In the case of the barrel shell shown in Figure III-3 Joints 50 and 51 are required to define the axis of the cylinder about which the joints are to be generated and Joints 1 and 43 are required to define the extent of the structure along the axis of the cylinder.

To define these starting joints do the following:

1. Since we would be adding joints in 3D space, check that the display is set to 3D in the Control Panel. If it is set to 2D, click on it once to change it to 3D.

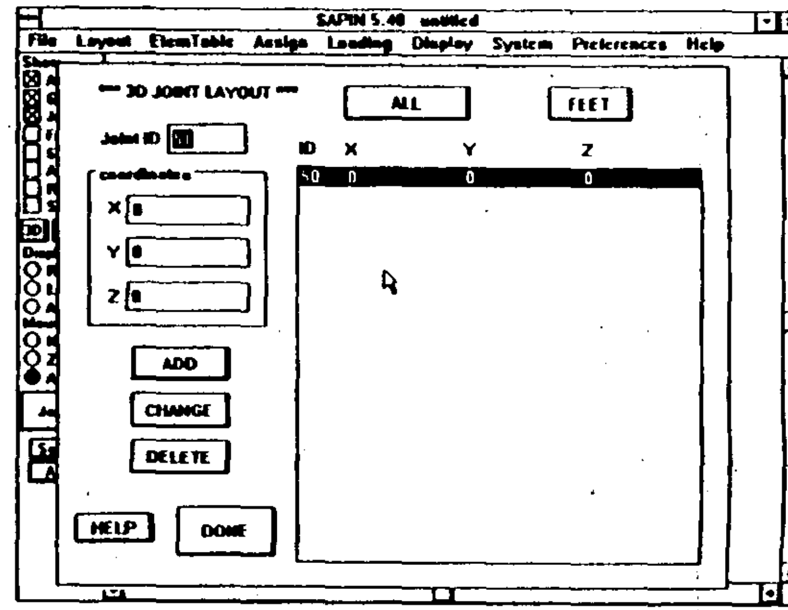
2. Click on **Layout** on the menu bar. Click on **Joints** in the **Layout** pull down menu. The 3D JOINT LAYOUT dialog box will appear as follows:



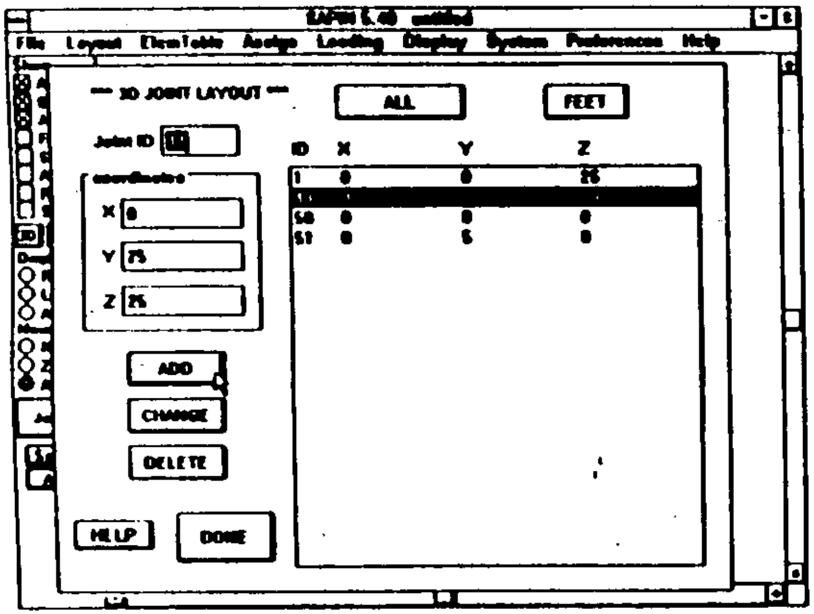
3. Click on the length unit push button to change it to FEET.

4. We will define Joint 50 first. Type in 50 in the edit box labeled **Joint ID**.

5. Since the default values (0,0,0) of the coordinates showing are correct for this joint, click on ADD. The program will add Joint 50 to the list of joints and increment the joint number showing in the edit box by 1 in anticipation that the new joint number will be defined next. The screen will show as follows:



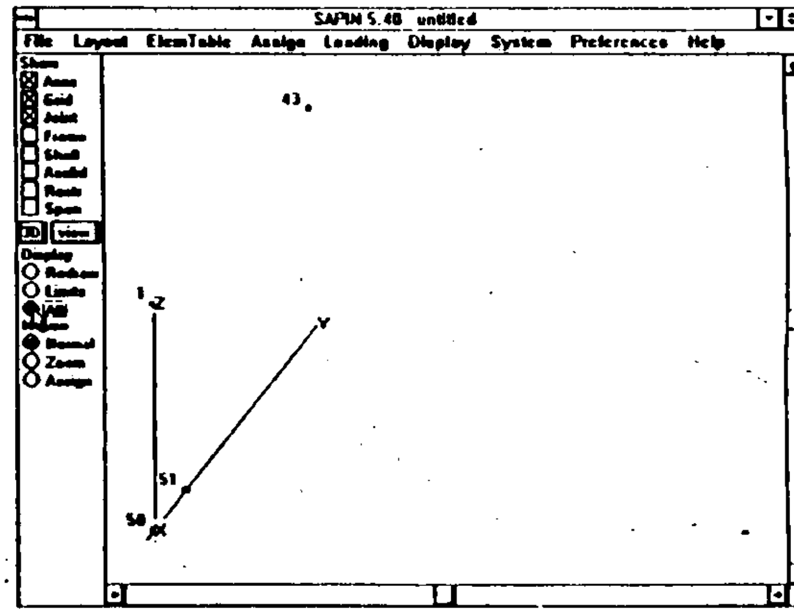
6. Repeat the above procedure to add Joints 51, 1 and 43 with the required coordinates of (0,5,0), (0,0,25) and (0,25,25). The screen will look as follows after these joints have been added:



7. Click on DONE. The program will close the 3D JOINT LAYOUT dialog box and the blank SAPIN window will appear.

8. To view the model click on **All** in the Control Panel and a model showing the defined joints will appear.

9. To show joint numbers, click on **Display Options** under the **Display** pull down menu. The DISPLAY OPTIONS dialog box will appear. Check **ID number** to be on under joints. Click on DONE to close the dialog box. Now click on All or Redraw in the Control Panel and the following screen will show:



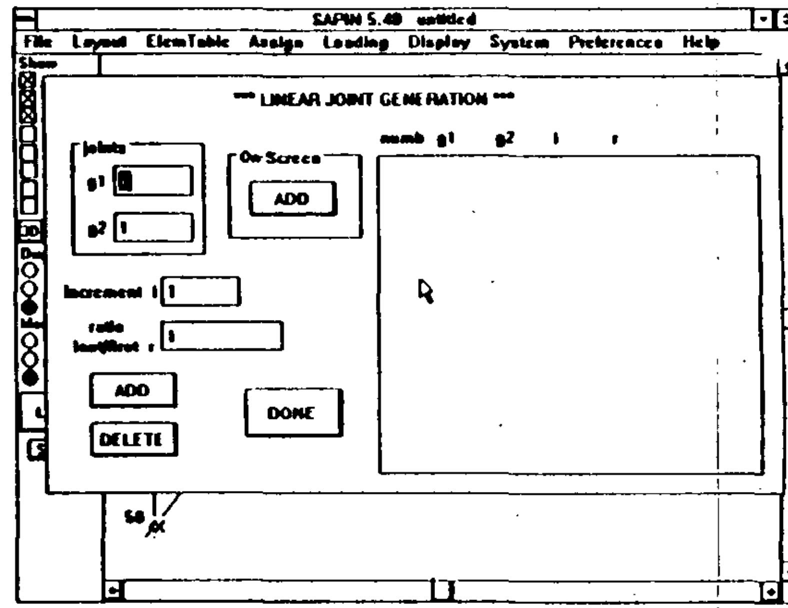
ii. Generating Joints

There are three different generations involved to complete the joint specification of this structure. A linear generation to define the joints along the line from Joint 1 to Joint 43; a cylindrical generation to define Joints 2 to 7 along the curve; and a frontal generation to complete the mesh.

There are also two choices available to generate joints. The first one is off screen in which all generation parameters and starting joint numbers are specified in the dialog box. The second one is on screen in which the generation parameters are specified in the dialog box but the starting joint or joints are picked on the screen. We will use the off screen method for the linear and cylindrical generations and the on screen method for the frontal generation.

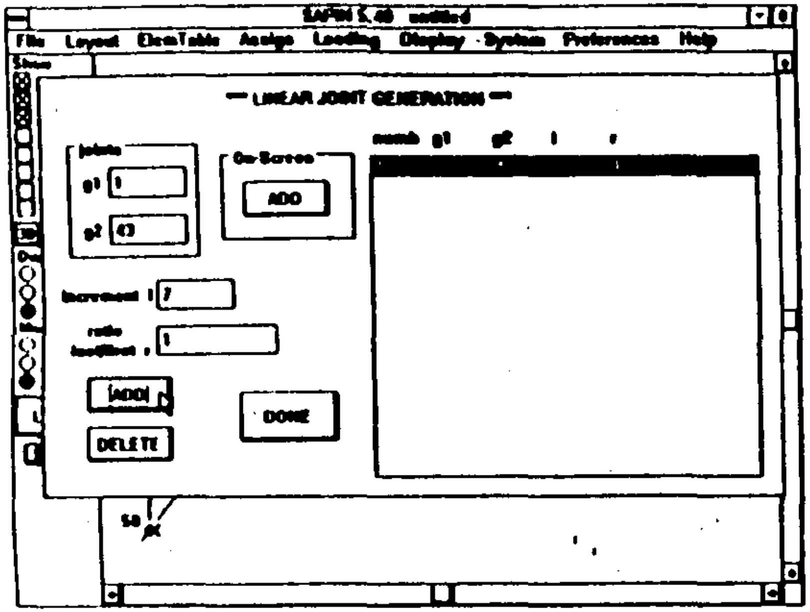
To generate the joints do the following:

1. Click on **Layout** on the menu bar. Click on **Linear** in the **Layout** pull down menu. The following Linear Joint Generation dialog box will appear:



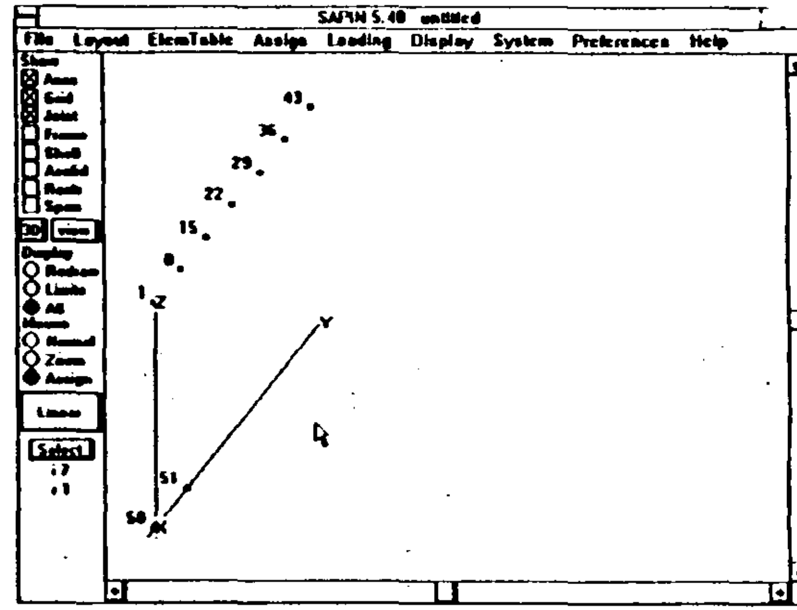
2. Type in **1**, **43** and **7** in the edit boxes labeled **joints g1**, **g2** and **increment i**, respectively. It is noted that the **ratio last/first r** is defaulted to 1. This is the ratio of the spaces between the first two joints and the last two joints. A ratio of 1 means equally spaced joints which we want in this example.

3. Click on ADD. The program will add this linear generation in the list and the screen will appear as follows:

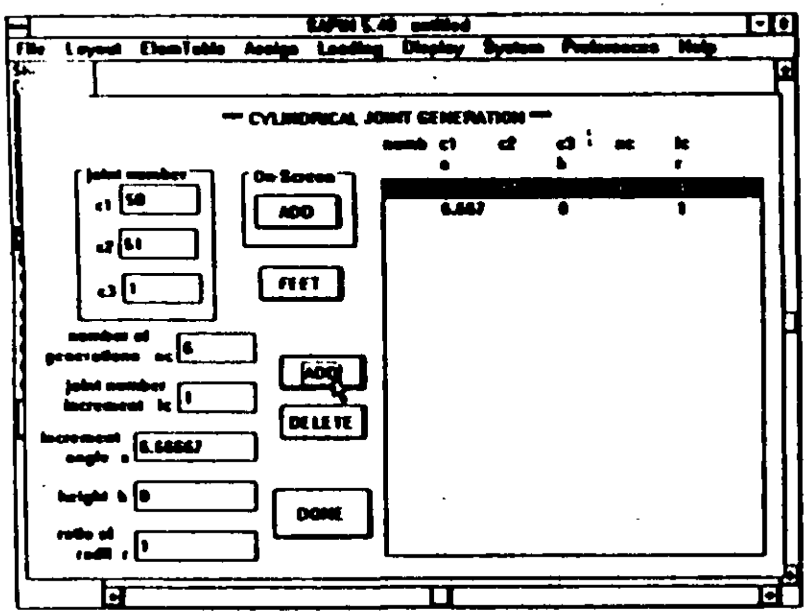


4. Click on DONE. The program will close the LINEAR JOINT GENERATION dialog box.

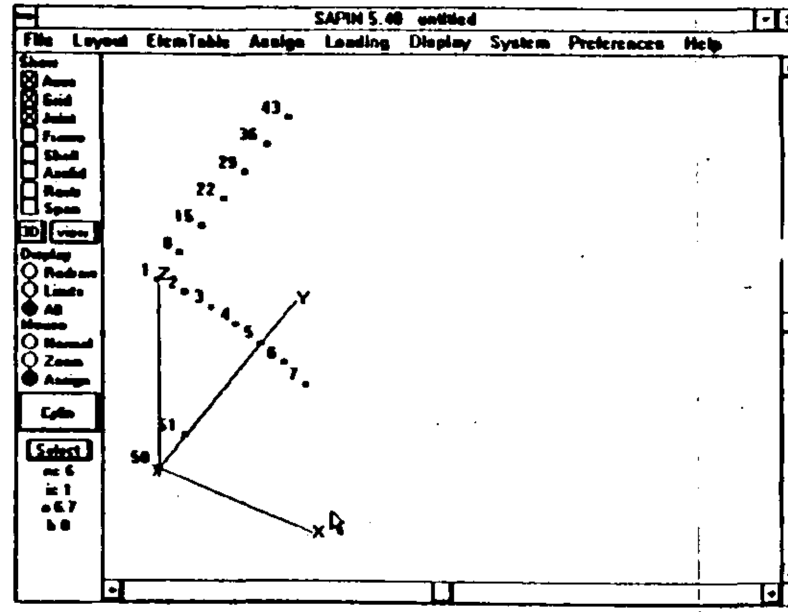
5. Click on All or Redraw in the Control Panel and the model with the currently defined joints will show as follows:



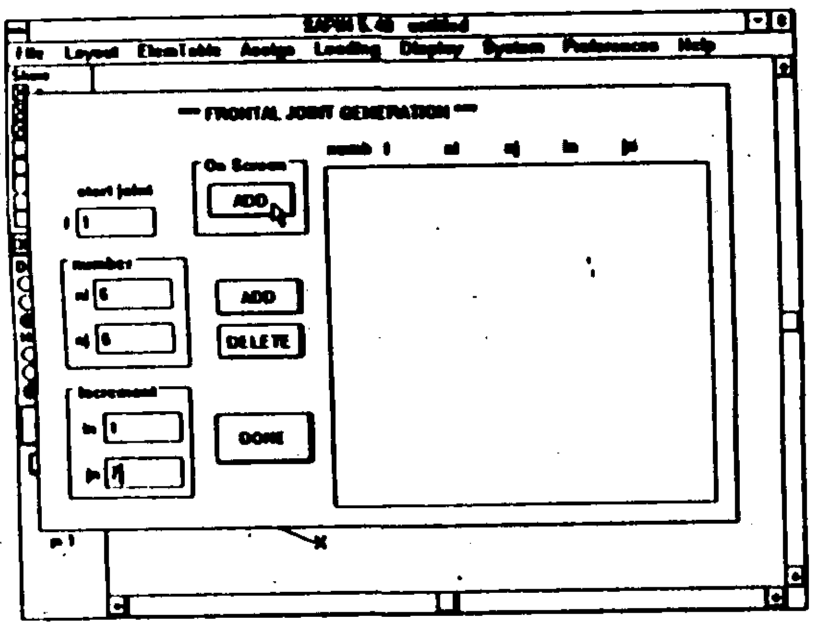
6. The cylindrical joint generation is similar to above. Select **Cylindrical** from the **Layout** pull down menu to open the CYLINDRICAL JOINT GENERATION dialog box. Type in **50, 51, 1, 6, 1** and **6.66667** (i.e. 40/6) in the edit boxes labeled **joint number c1** and **c2** defining the axis of the cylinder, **c3** the starting joint, **number of generations nc**, **joint number increment ic**, and **increment angle a**, respectively. It may be noted that the edit boxes for **height h** and **ratio of radii r** are left with the default values of 0 and 1, respectively. Other values for these parameters are used to generate joints along spirals and helices. Click on ADD. The program will add the cylindrical generation to the list and the screen will show as follows:



7. Click on DONE to close the dialog box. Now click on **All** or **Redraw** in the Control Panel, the program will show the model with the currently defined joints as follows:

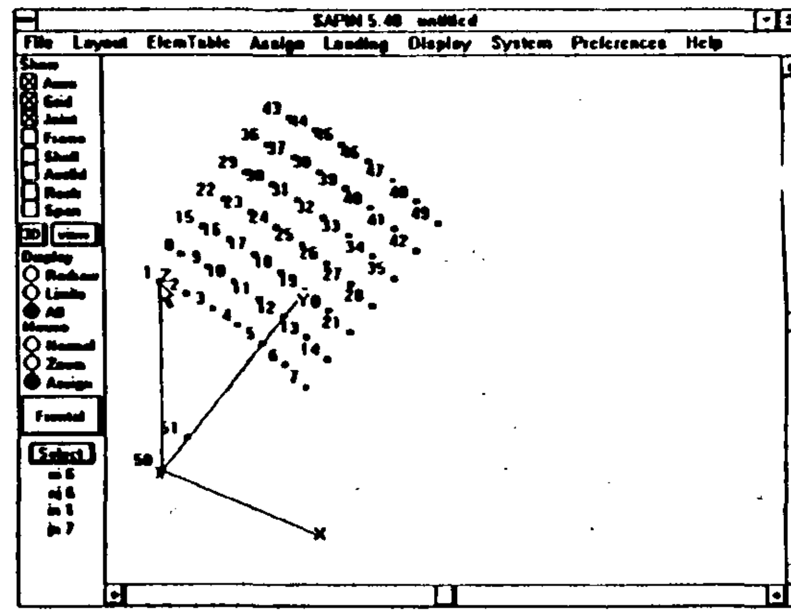


8. The frontal generation is also similar, except we will use on screen generation. Select **Frontal** from the **Layout** pull down menu to open the FRONTAL JOINT GENERATION dialog box. Type in **6, 6, 1** and **7** in the edit boxes labeled **number ni** for the number of additional joints in the i direction, **nj** for the number of additional joints in the j direction, **increment in** for the joint number increment in the i direction and **jn** for the joint number increment in the j direction, respectively. The screen will appear as follows:



9. Click on ADD in the On-Screen box. The program will now show the model with the currently defined joints.

10. Click on Joint 1. The program will generate the additional joints to complete the mesh and the screen will appear as follows:



The joint generations are complete.

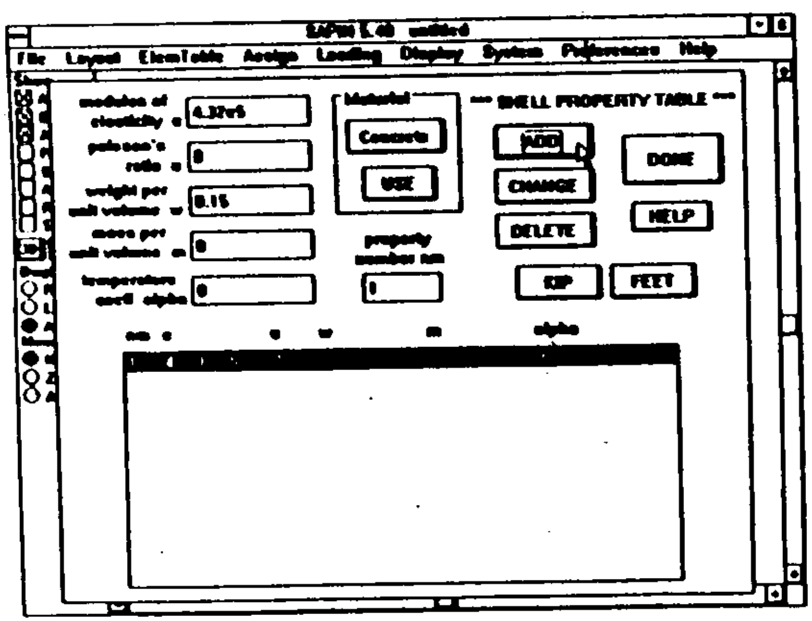
iii. Defining Shell Element Material Properties

To define shell element material properties do the following:

1. Click on **ElemTable** on the menu bar. Click on **Shell** in the **ElemTable** pull down menu. The SHELL PROPERTY TABLE dialog box will appear.

2. Type in **4.32e5** and **.15** in the edit boxes labeled **modulus of elasticity e** and **weight per unit volume w**. Note that the units are KIP and FEET.

3. Click on ADD. The program will add this material property to the list of properties and the screen will appear as follows:



4. Click on DONE. The program will close the SHELL ELEMENT TABLE dialog box and the program will return with the earlier SAPIN window.

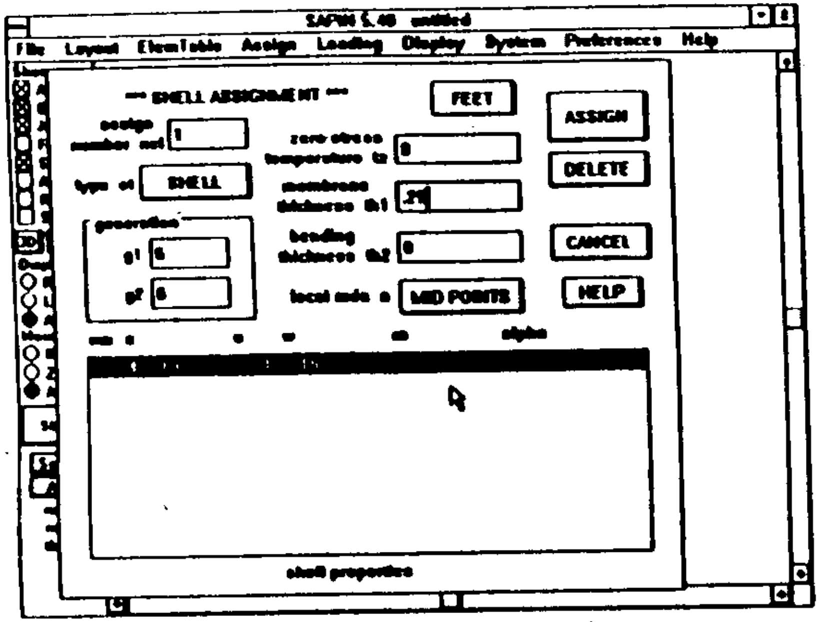
iv. Locating Shell Elements

All elements can be assigned one at a time or can be generated. For the elements to be generated it is important that the joint numbering follow certain regular patterns. Since the joint numbering for this example is regular we will use the element generation option.

To locate (assign) shell elements between the joints do the following:

1. Click on Assign on the menu bar. Click on **Shell** in the **Assign** pull down menu. The SHELL ASSIGNMENT dialog box will appear.

2. Type in **6**, **6** and **.25** in the edit boxes labeled **generation g1** for total number of elements in the 1 direction, **g2** for total number of elements in the 2 direction and **membrane thickness th1**, respectively. The other edit boxes and push buttons are left as defaults. The **assign number nel** of 1 means the starting element ID number will be 1. The generated elements always get ID numbers incremented by 1, first in the 1 direction then in the 2 direction. The **type et** of SHELL means the shell to be assigned has both membrane and bending stiffness. The **zero-stress temperature tz** of 0 means the reference temperature for zero thermal stress is 0. The **bending thickness th2** of 0 means default the bending thickness of the shell to be equal to the membrane thickness. The **local axis n** of MID POINTS means the local shell axes for outputing the stress results should be based on the mid-points of the element edges. The screen will now show as follows:

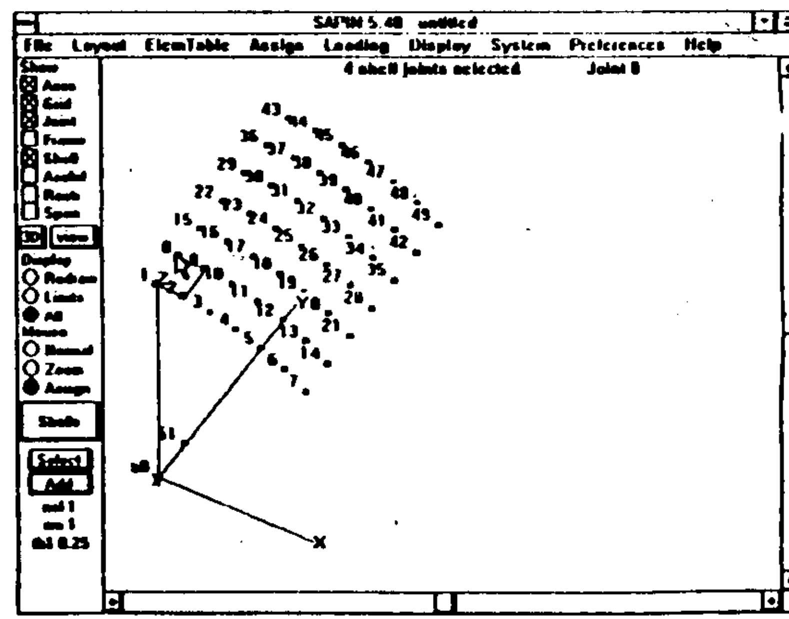


3. Click in the list box on the material property to be assigned.

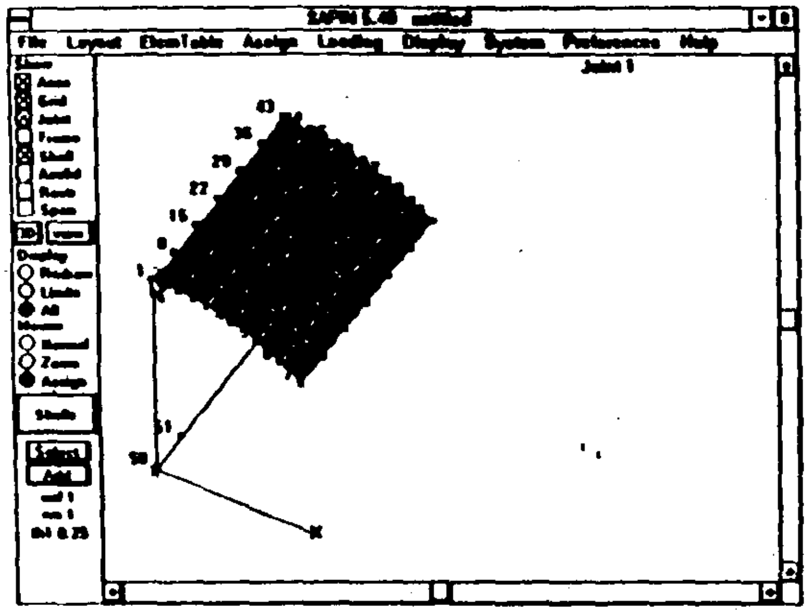
4. Click on ASSIGN. The program will close the SHELL ASSIGNMENT dialog box and the current model of the structure with the joints showing will appear on the screen.

5. To assign a shell element we need to click on the joints to which it is connected. The clicking should be done on the joints going either clockwise or counter-clockwise. Counter-clockwise would direct the normal to the shell out of the screen. Clockwise would direct it into the screen. This is important when pressure loads are applied. Also the direction defined by the first two joints clicked is the direction in which **g1** elements will be generated.

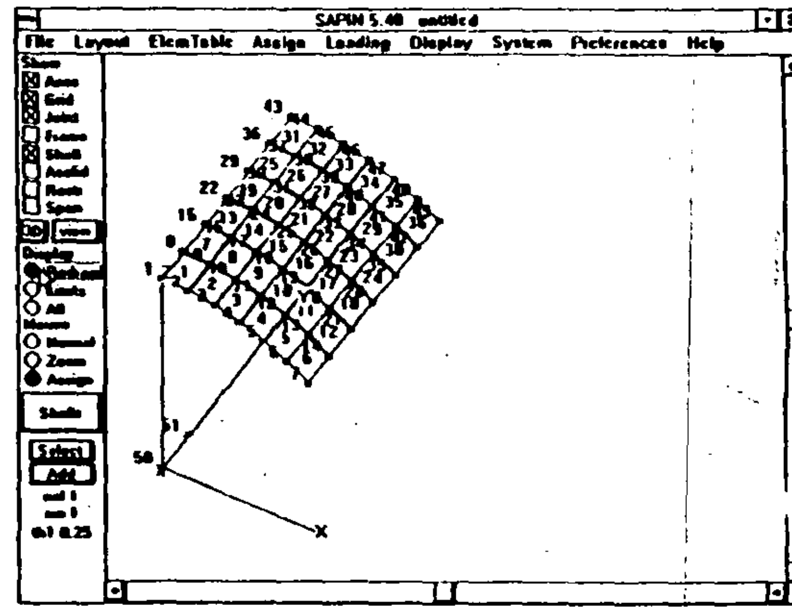
To assign the starting element for our problem, click on Joints 1, 2, 9, 8 and 1 (again). Notice that a message on the right top of the screen shows the joint number clicked on. Also a message at the center top of the screen shows how many joints have been clicked in this sequence. The program also draws a line between the joints, showing the outline of the element being assigned. After the first four joints have been clicked, the screen will look as follows:



6. As soon as Joint 1 is again clicked on the assignment of the starting element is complete and the program will automatically generate the requested number of elements and the screen will look as follows:



7. The default is to fill the shell elements and not to show their ID numbers on the screen. To view the ID numbers, click on **Display Options** in the Display pull down menu. The DISPLAY OPTIONS dialog box will show. Check **ID number** under shell to be on and click on **outline** under shell to turn it on instead of filled. Click on DONE to close the dialog box. Now click on **Redraw** in the Control Panel. The following screen will show:



The shell assignments are now complete.

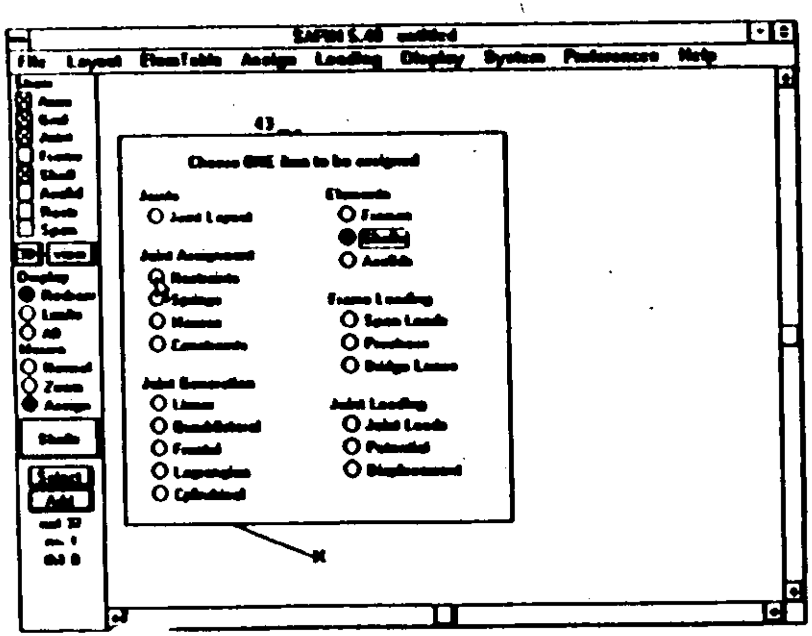
v. Assigning Restraints

Three different types of restraints are required for this example. The simply supported restraint at the curved edge and two different types of restraints to model the two symmetry conditions.

We can access the restraint assignments either by selecting **Restraints** under the **Assign** pull down menu, or by selecting restraints through the Control Panel. For this example we will select restraints through the Control Panel.

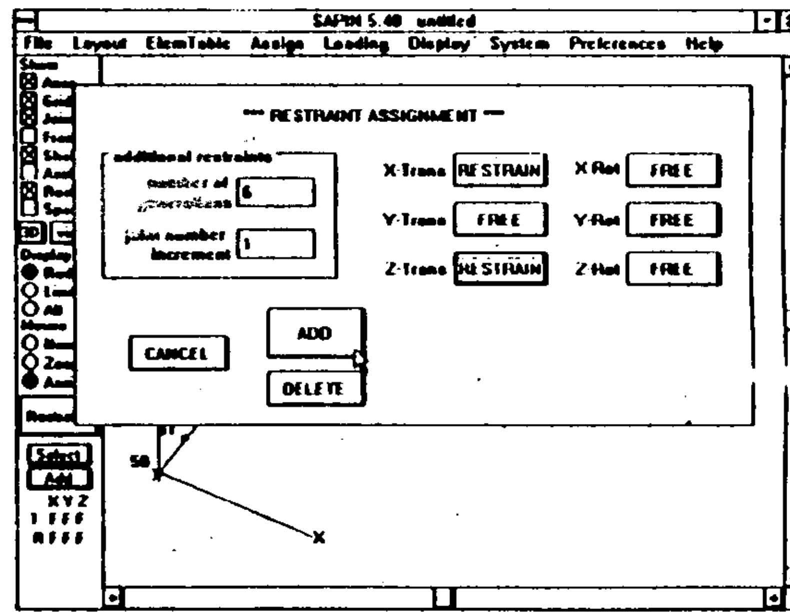
To specify restraints do the following:

1. Click on the pushbutton immediately under the **Assign** button in the Control Panel. If nothing is showing under the Assign button, click on it first. The following assign selection box will appear.



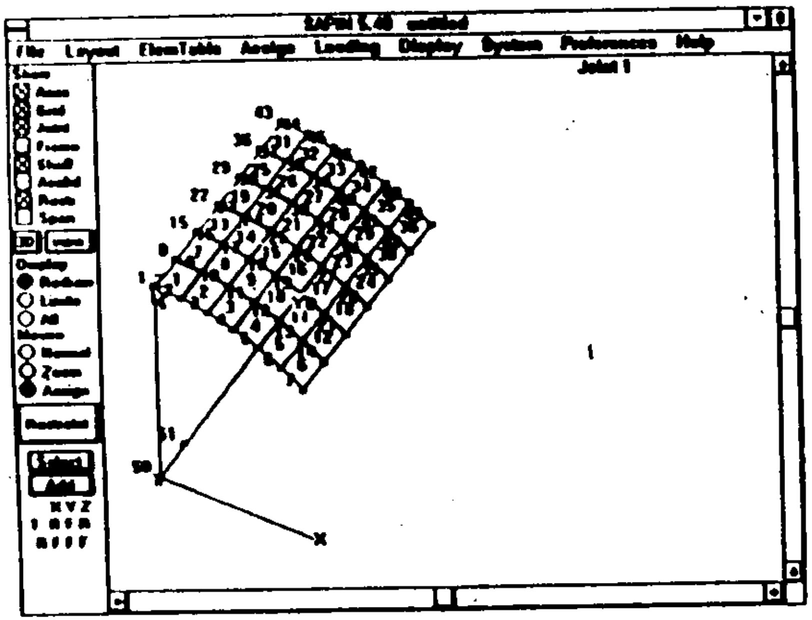
2. Click on **Restraints**. The assign selection box will close and the program will be ready to assign restraints as shown by the button under Assign in the Control Panel. However, the default restraints will fix all degrees of freedom at the joint. To change to a different set of restraints, click on **Select** in the Control Panel. The RESTRAINTS ASSIGNMENT dialog box will appear.

3. To model the simply supported restraints at the supported curved edge; type in **6** in the edit box labeled **number of generations** for the restraints to be generated at these number of additional joints and set the restraints push buttons to the following settings;

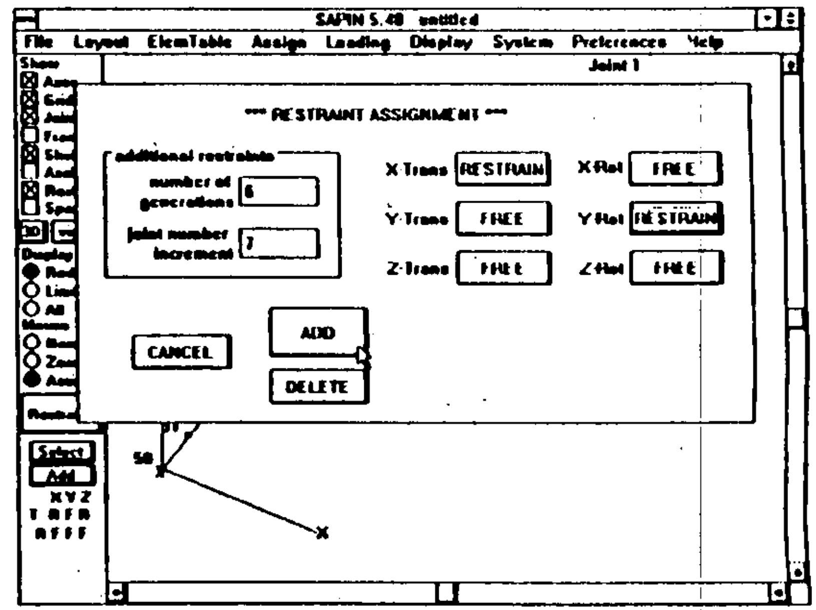


4. Click on ADD. The RESTRAINT ASSIGNMENT dialog box will be closed and the current model of the structure will appear on the screen.

5. Click on Joint 1. The program will assign the restraints from Joint 1 to Joint 7 and the color of these joints will change confirming that. The screen will look as follows:

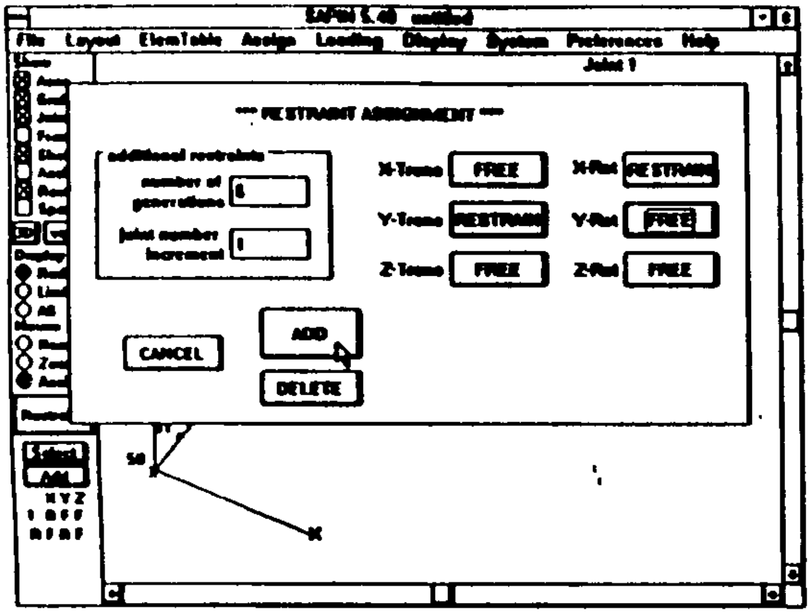


6. To model the line of symmetry along the top of the structure, click on Select in the Control Panel. The RESTRAINT ASSIGNMENT dialog box will reopen. Type in **6** and **7** in the edit boxes labeled **number of generations** and **joint number increments**, respectively. Also set the restraints as shown in the following screen:

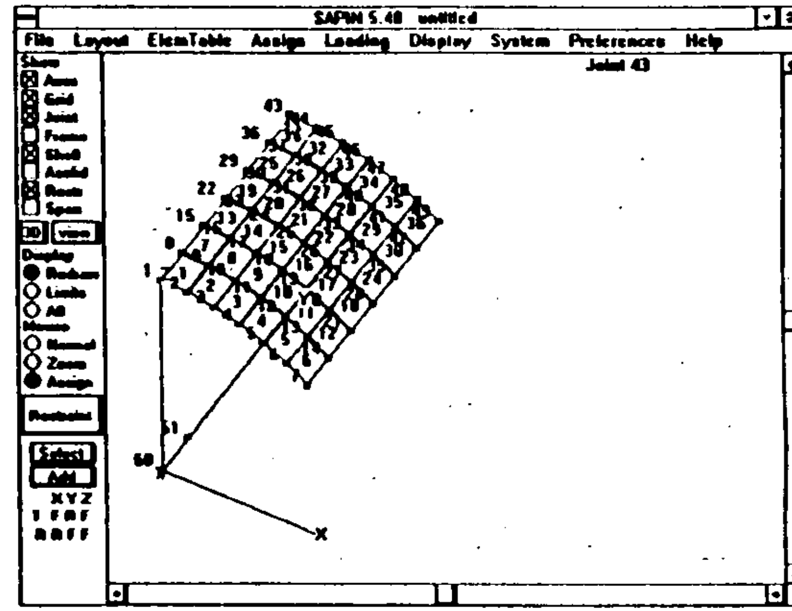


Click on ADD to close the RESTRAINT ASSIGNMENT dialog box and then click on Joint 1 to assign the restraints. The top line of joints will change color indicating the assignment of the restraints.

7. To model the line of symmetry along the curved edge in the center of the structure, repeat the above procedure except the RESTRAINT ASSIGNMENT dialog box should be initialized as shown on the following screen:



Also, the restraint should be assigned to Joint 43 as shown on the following screen:



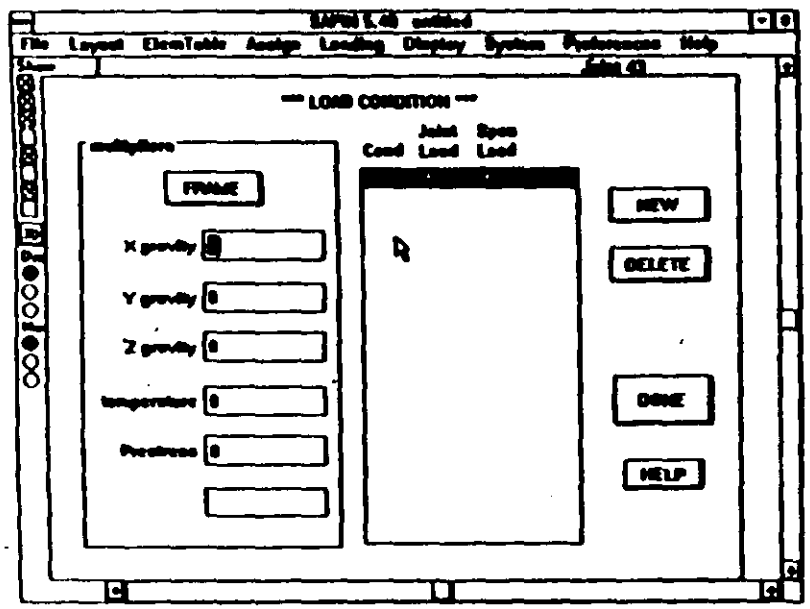
The joint restraint specifications are now complete.

vi. Assigning Loading

The distributed vertical load on the surface of the shell will be modeled as a gravity multiplier for the self weight of the shell. The multiplier to model a 90 psf load for a 3 inch thick shell of weight density 150 psf will be -2.4 (i.e. 90/(150*3/12)). The negative sign is for the load to act vertically downwards (-Z direction.)

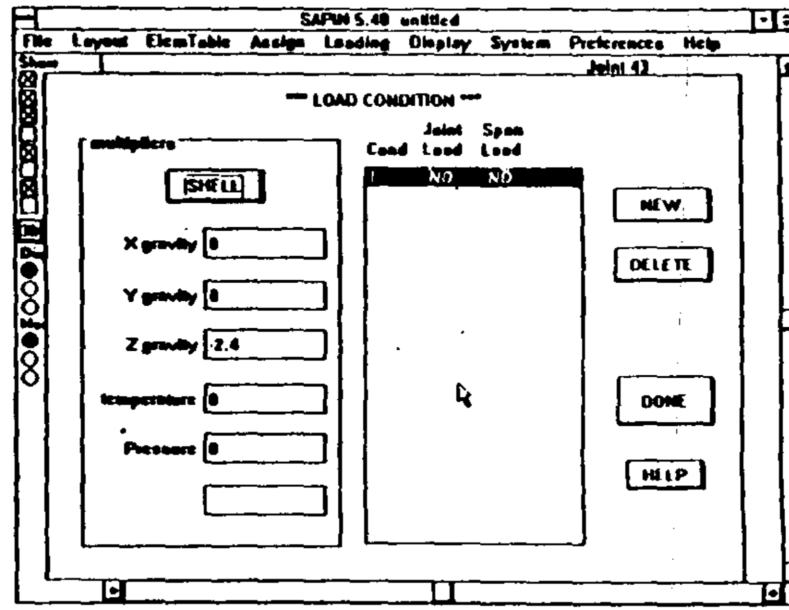
To specify the above load do the following:

1. Click on **Loading** on the menu bar. Click on **Load Condition** in the **Loading** pull down menu. The following LOAD CONDITION dialog box will appear:



3. Click on the push button in the MULTIPLIERS box to change it to SHELL from FRAME.

4. Type in -2.4 in the edit box labeled Z gravity. Note that this multiplier is being assigned to load condition 1 as that is the one highlighted. (To assign loads to other load conditions, they should be added first using the NEW push button and then selected from the list before any multipliers are entered or joint or span loads assigned.) The screen will now look as follows:



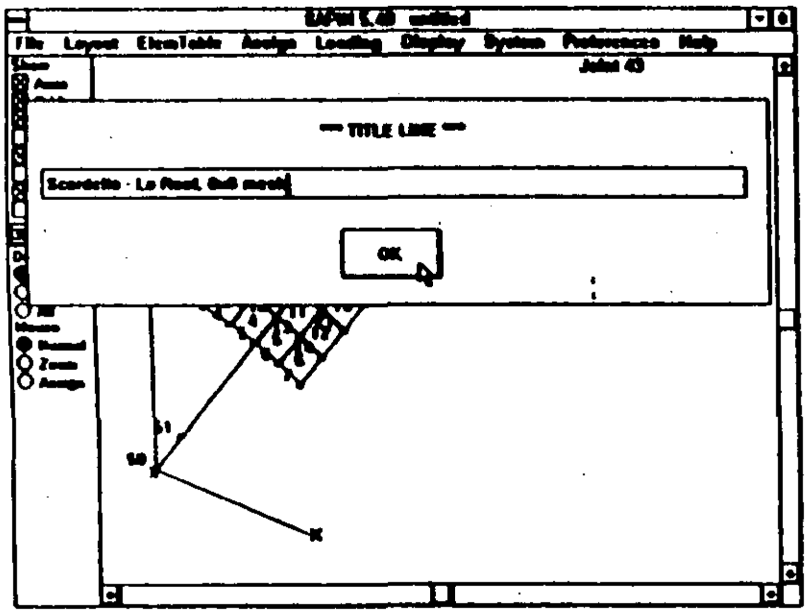
5. Click on DONE. The program will now close the LOAD CONDITION dialog box.

The load assignment is now complete.

vii. Saving the Model

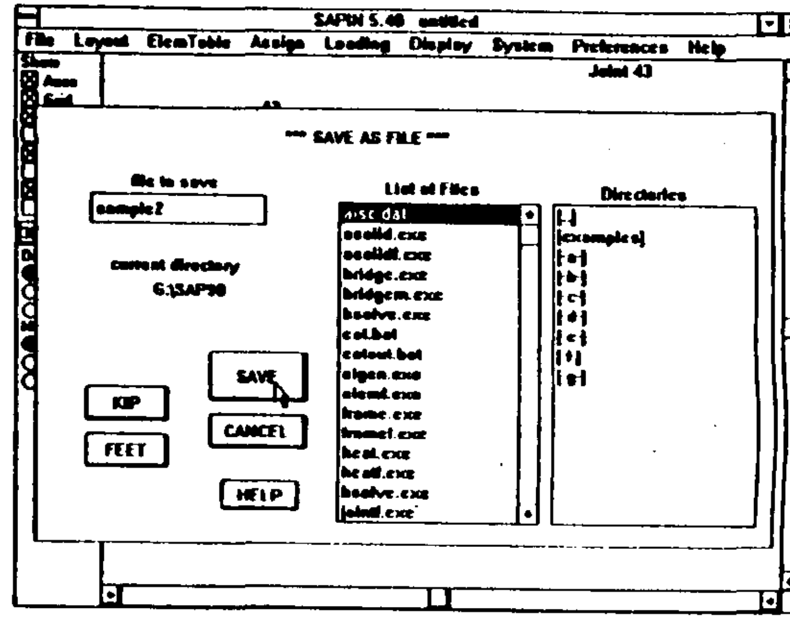
The saving of the model is identical to Exercise One. A few additional features are introduced here.

1. A title line may be added to the SAP90 file created by SAPIN. To do this click on **Title Line** in the **File** pull down menu. The TITLE LINE dialog box will appear. Type in the title as shown in the screen below:

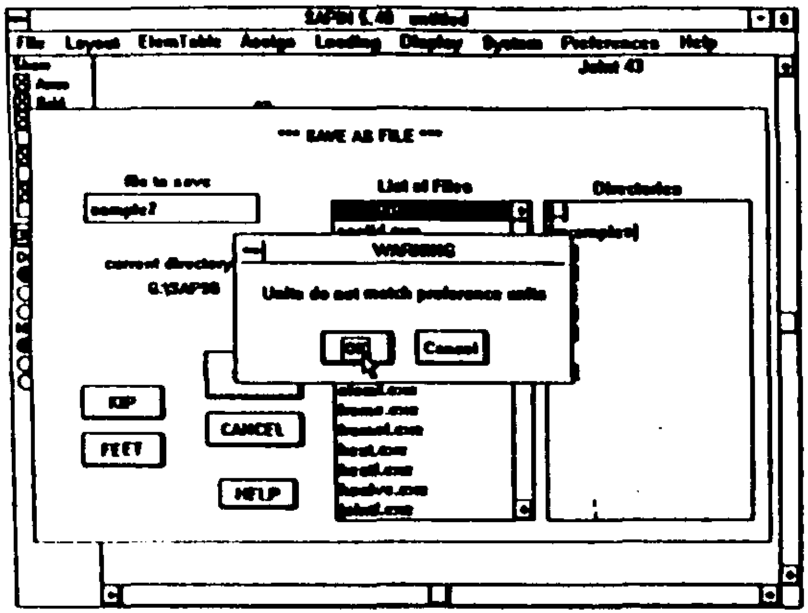


Now click on OK. The program will close the TITLE LINE dialog box.

2. To save the file, click on Save As in the File pull down menu. Type in **sample2** in the file to save edit box. The screen will show as follows:



3. Noting that the units, filename and directory are as we wish, click on SAVE. The program will issue a warning message as shown below:



4. Click on OK. The program will now close the warning window and the SAVE FILE dialog box and the model will be saved in an ASCII file called SAMPLE2. Since we have saved the file in other than the preference units, we should be careful that the units in which this file is opened later for any modifications are consistent with the units in which it was saved.

If the file is viewed on the screen or printed it will appear as shown in Figure III-4.


```
Scottsdale - Lo Roof, 6x6 mesh
C This is file SAMPLE2 written by SAPIN on Tue Mar 10 14:02:29 1992
C Units are KIP FEET
SYSTEM
 R=0 L=1 C=0 V=0 T=0.0001 P=0 W=0 Z=0
JOINTS
 50      X=0 Y=0 Z=0
 51      X=0 Y=5 Z=0
 1       X=0 Y=0 Z=25
 43      X=0 Y=25 Z=25
 1       X=0 Y=0 Z=25 G=1,43,7,1
 50      X=0 Y=0 Z=0 A=50,51,1,6,1,6.6667,0,1
 1       X=0 Y=0 Z=25 F=1,6,6,1,7

SHELL
 NM=1 Q=0 Z=-2.6
 1   E=4.32E+05 U=0.15 W=0 TA=0
 1       JQ=1,2,8,9 ETYPE=0 M=1 TZ=0 TH=0.25,0 LP=0 G=6,6

RESTRAINTS
 1    7    1    R=1,0,1,0,0,0
 1    43   7    R=1,0,0,0,1,0
 43   49   1    R=0,1,0,1,0,0
```

File SAMPLE2
Figure III-4

IV.

REFERENCE

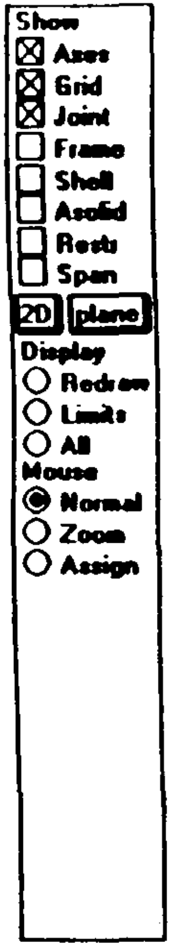
The reference section provides a detailed explanation for the Control Panel and each of the menu items. The Control Panel is described first, followed by the menu items in the order that they appear in the menu bar. Before we start, let's look at Help and Units.

Help

Most of the dialog boxes in SAPIN have HELP push buttons. Click on these to display an abbreviated help message. The help messages (and this manual) provide assistance for the use of SAPIN only and do NOT provide assistance for structural design and the definition of structural terms used in SAP90 and BRIDGE. Please refer to those manuals for assistance with structural items.

Units

Many of the dialog boxes have Units push buttons. The options for the force units are kip, pound, kilogram (force) and kiloNewton. The options for the length units are inches, feet, meters and millimeters. Click on these buttons to change units as desired, but be aware that the units are changed in the ENTIRE program. There are many cases where changing units will make entering data easier, but be sure to make the units consistent when Opening and Saving files. Saving a file in Kip-inches (for example) and then reading it back in Pound-feet will produce data that you may not recognize !



Control Panel

The Control Panel always appears at the left edge of the screen. It is used to control what is shown on the screen, when the screen is drawn, how the mouse operates and it is also used to assign structural elements and joints without going through the menus.

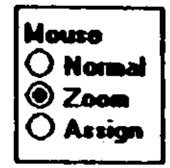
The Control Panel is a dialog box, exactly like all other dialog boxes, except that it is always visible. It contains check boxes, radio buttons and push buttons, all described in Chapter I. There is NO way to enter data in the Control Panel.

The Display group of radio buttons is used to draw the structure on the screen. What is drawn is controlled by the Show group of check boxes. The Mouse group of radio buttons controls the operation of the mouse.

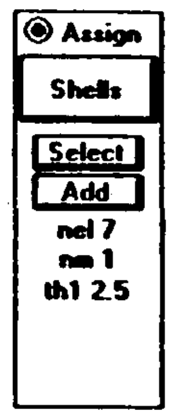
Assuming that you have a grid, joints and some structural elements, they will ONLY be drawn on the screen when you click on one of the Display radio buttons. The same button can be clicked repeatedly, if desired. All draws the entire structure and grids, Limits draws the portion of the structure that is set in Display Limits under the **Display** menu and Redraw draws the same thing that is on the screen along with any changes that may have been made.

Items that have been checked in the Show group of check boxes will be drawn if they exist, and items that are not checked will not be drawn. In this way, you can see frames alone, for example.

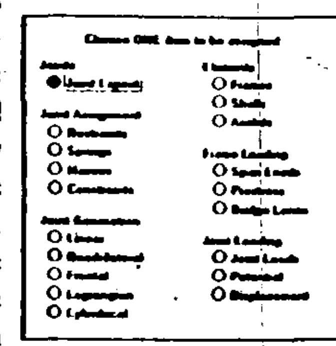
The Mouse group of radio buttons allows you to Zoom (magnify a portion of the structure) or to Assign joints, structural elements and loads.



Clicking on the Zoom button puts the mouse into Zoom mode, which is described in Chapter II. You can Zoom repeatedly for greater magnification. To return to the full view, click on All or Limits. The scroll bars at the right and bottom of the screen allow you to pan across the structure while retaining the same magnification. The scroll bars have no effect unless the view is magnified by zooming. Once you are in a Zoomed view, use Redraw to redraw the screen without changing magnification.



Clicking on the Assign button puts the mouse into assign mode, which is described in Chapter II. When assign is selected, then the space under the Assign button is used to display information about the item to be assigned and also to allow you to change the assignment type. As shown at the left, there is one large push button under the Assign radio button which shows the type of item being assigned. Clicking on this button brings up a dialog box as shown at the right. Click on one of the radio buttons to select the item to be assigned.



In the example shown, clicking on the Select button brings up a small dialog box allowing you to change the most used shell assign items. The Add pushbutton can be changed between Add and Delete. The 3 remaining items show the present values of nel (assign number), nm (property number) and th1 (thickness). These 3 items can be changed using the Select button.

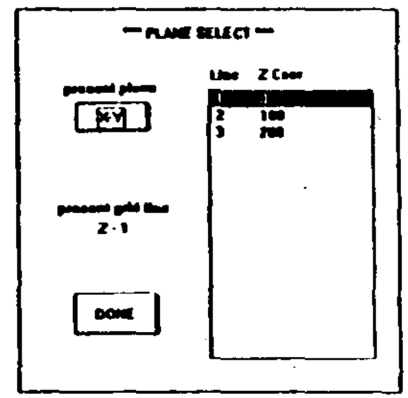
A complete description of the items shown is given later under the menu selection for the item assigned. In this example, look at Shell under the **Assign** menu later in this chapter.

Please be aware that you must do some preliminary work before the Control Panel is of any use. In this example, you must have previously defined joints on which to assign the shell, at least one shell property must have been defined, and probably you will want to check the items under Shell in the **Assign** menu. The purpose of the Control Panel is simply to assist you in assignment by showing you abbreviated information about the item assigned and to allow changing the most common items.

2D and 3D Display

The display can show either a 2D or 3D view of the structure. To show a 2D view, you MUST have defined X, Y and Z grids previously. The type of display is controlled by a push button in the Control Panel. Clicking on 2D switches to 3D and clicking on 3D switches to 2D.

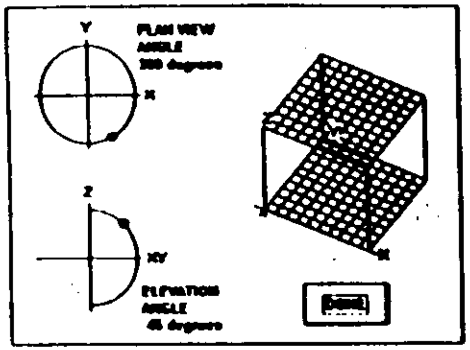
When in 2D display, clicking on the plane push button brings up the following dialog box:



This dialog box allows you to select the plane that will be displayed on screen. If the list box is empty, there are no grids defined and nothing will be displayed.

Click on the present plane button to set the axes, then click on the desired plane in the list box and finally click on DONE. The selected plane will be displayed.

When in 3D display, clicking on the view button will bring up this dialog box:

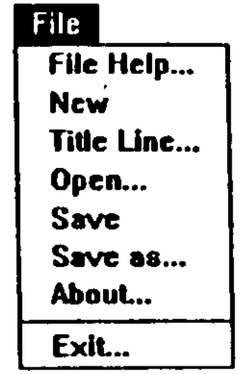


Note that if there are no grids defined, then the grids at the right of the dialog box will not be shown. One of the advantages of grids is that they will graphically show the view direction in this dialog box (when you change the view angles, the grid view will change to match). Without grids, you will have to rely on the view angles only.

This dialog box allows you to select the view direction for the 3D display. Point to one of the dots on the circles, PRESS and HOLD the left mouse button and drag the dot around the circle until it is at the angle you want. Once you have selected the dot, you can move the mouse cursor away outside the circle to make it easier to get an exact angle.

This entire process is made much easier if you have defined grids, as the top and bottom grids will be shown at the right and when you release the mouse button, the grids will be drawn in the correct view. Defining grids with only 2 grid lines each is helpful here, even if you don't use them for anything else.

When you click on DONE, the selected view will be displayed.



The **File** menu allows you to define, edit and save the files that SAPIN uses. ALL files are "SAP90 INPUT DATA FILES" as defined in Chapter IX of the SAP90 Users Manual. These files are in the standard ASCII format and may be edited using any text editor in ASCII mode. DO NOT use a text editor in non-ASCII mode as it will put characters in the file that can not be recognized by either SAPIN or SAP90. Following is a definition of **File** commands:

File Help

Clicking on **File Help** brings up a dialog box which contains abbreviated information on using the **File** menu.

New

Clicking on **New** resets SAPIN to the state it is in when started. If there is any structure defined, it will be erased, including all elements and assignments. Further, capacity, materials, units, colors and directory are set from the file SAPIN.INI (if there is one).

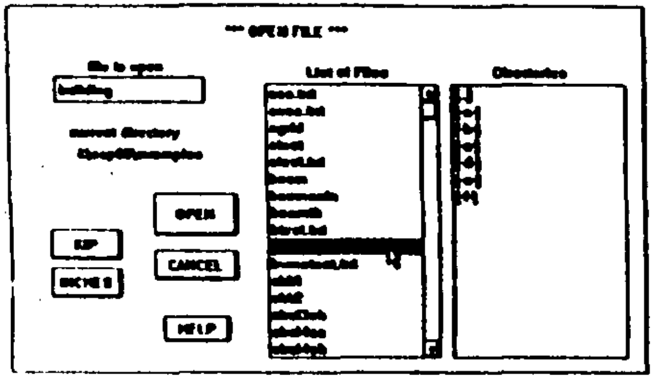
Title Line

The first line of every SAP file can contain up to 70 characters with any information you desire. Click on **Title Line** and move the cursor to the edit box and type in what you want. When you do **Save** or **Save as**, the current title line will be written to the file. Any characters past 70 will not be written.

Open

Open allows you to read in the current structure from an existing file. The file becomes the current file and its name appears at the top center of the screen.

When you click on **Open**, this dialog box appears:



First select the proper Current Directory from the Directories list box. The items in this box are drives or directories. For example, [-A-] is drive A: and [Name] is directory Name. Clicking on a drive will make the root directory of that drive the current directory. Clicking on a directory will make it the current directory. There is the special case of [..] which means the next higher directory level, just as it does in DOS. Clicking on [..] will make the next higher directory the current directory. If the root directory is the current directory, then there is no higher level directory and [..] will not appear.

With the correct current directory selected, you can now select a file by clicking on it in the list of files, or alternatively, move the cursor to the File to Open edit box and enter the desired name. Then click on the OPEN button to open the file. You can shortcut this procedure by just double-clicking on the file name in the list box.

The current structure will be read in from the file, with the units specified, OVERWRITING any existing structure.

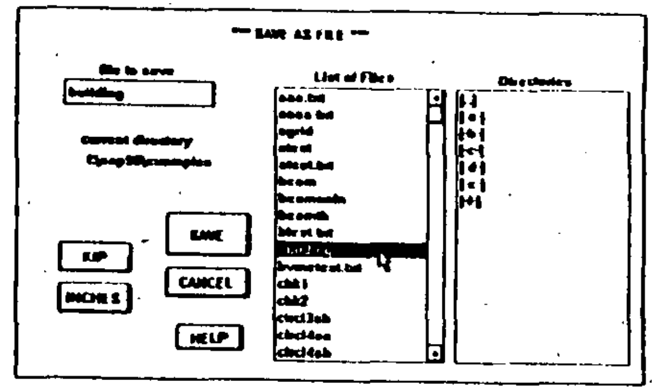
Save

Clicking on **Save** OVERWRITES the current file (name at top of screen) with the current structure. The file will be saved in the units that are currently set. If these units do not match the preference units, you will be allowed to cancel the save if desired. It is not necessary that the units match the preference units, this is just a warning.

This is the easy way to periodically save your work, but it is a good idea to change the file name using **Save as** every so often to avoid losing your work.

Save as

Save as allows you to save the current structure in a file whose name you specify. When you click on **Save as**, the following dialog box appears:



First select the proper current directory in exactly the same way as described for **Open**.

Then select a file by clicking on it in the List of Files, or if it is not in the list, move the cursor to the File to Save edit box and enter the desired name. Then click on the SAVE button to save the file. Double clicking on the file name in the list of files will also save the file.

The current structure will be written to the file, which becomes the current file. If the file exists, you can choose to OVERWRITE the file with the current structure or to cancel the save.

The file will be written in the units specified, regardless of the units you may have been previously using. If the units do not match the preference units, you will be allowed to cancel the save if desired.

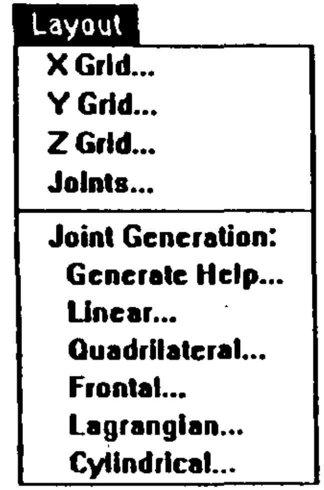
About

Clicking on **About** displays a dialog box which shows copyright information and the size of available memory. If you are running in Windows 386 Enhanced mode, the memory size will be larger than the actual memory you have, because the disk is used for the extra memory. You can run Windows in Standard mode (by starting Windows with Win /S) to determine the actual amount of free memory.

If your structure is large enough to use all of memory and you are in 386 Enhanced mode, then the disk will be used as you increase the structure size. The effect of this is a noticible drop in speed. The cure is to add more memory.

Exit

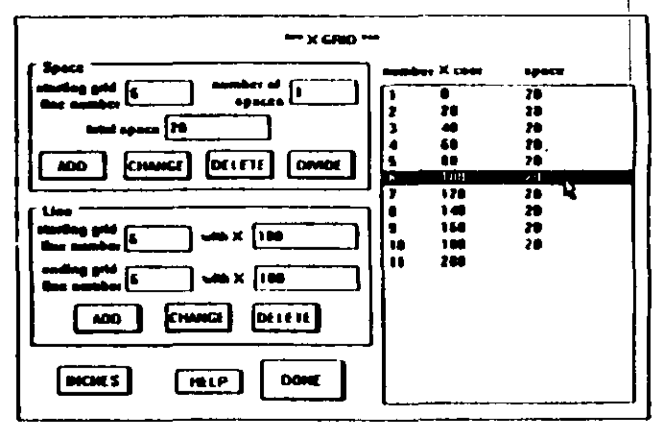
Clicking on **Exit** has the same effect as double-clicking on the close box in the upper left hand corner of the screen, that is, it stops SAPIN. You will be given a chance to **Save** your structure before you exit, as otherwise, ALL information entered after the last **Save** will be lost.



The **Layout** menu is where you define the joint layout of the structure. Since all structural elements are assigned to joints, the joint layout is the basic framework of the structure. X, Y and Z grids can be defined, which allows placement of joints on screen at grid intersections using the mouse (in 2D display only). Joints can also be defined by coordinates, and multiple joints can be generated in various patterns, such as rectangular and circular areas. Note that there are NO structural elements defined here, but only the framework on which structural elements are assigned later.

X Grid

Clicking on **X Grid** brings up the following dialog box:



The X grid lines are perpendicular to the X axis and cross it at the X coordinate specified for each line. Coordinates can be any value, plus or minus. You can deal with grid lines either by the spaces between lines or by the coordinates of the lines. Generally speaking, it is easier to define groups of lines by using spaces and then move or modify individual lines using coordinates, but every situation is different.

Using Spaces

To add new spaces, enter the starting grid line number, number of spaces, total space and click on ADD. If there are existing grid lines, then the starting grid line must be one of those lines. The spaces will be added AFTER the starting line and BEFORE the space between the starting line and the next existing grid line. This changes the line numbers and coordinates of any existing lines after the inserted spaces. If there are no existing grid lines, then the spaces will start with 0.0 coordinate.

To change the value of an existing space, enter the line number of the line BEFORE the space as the starting grid line number and enter the new space value as the total space. Then click on CHANGE. The number of spaces is not used. This changes the coordinates of existing lines above the space, but not their line numbers.

To delete one or more consecutive spaces, enter the line number of the line BEFORE the spaces as the starting grid line number and enter the number of spaces. Then click on DELETE. The total space is not used. This changes line numbers and coordinates of existing lines above the deletion.

To divide an existing space into 2 or more spaces, enter the line number of the line BEFORE the space as the starting grid line number and enter the number of spaces desired, (which must be 2 or greater). Then click on DIVIDE. This changes the line numbers but not the coordinates of existing lines above the division.

Using Lines

To add new lines, enter the starting grid line number and its coordinate, then enter the ending grid line number and its coordinate. Click on ADD to add the lines. If there are existing lines, then the new lines will be inserted AHEAD of the starting grid line number and the coordinates must be BETWEEN the two existing lines surrounding the insertion. Added lines will be evenly spaced. To add a single line, set the starting and ending grid line numbers to the same value. The ending grid line coordinate is not used in this case. Adding grid lines will change the line numbers but not the coordinates of existing lines above the addition.

To change the coordinate of a single line, enter its line number as the starting grid line number and its coordinate. Then click on CHANGE. The coordinate must be BETWEEN the coordinates of existing grid lines surrounding the change line. The ending grid line number and its coordinate are not used. This does not change other existing lines in any way.

To delete one or more consecutive existing lines, set the starting grid line number and the ending grid line number. Then click on DELETE. The coordinates are not used. This changes the line numbers but not the coordinates of existing lines above the deletion.

Y Grid

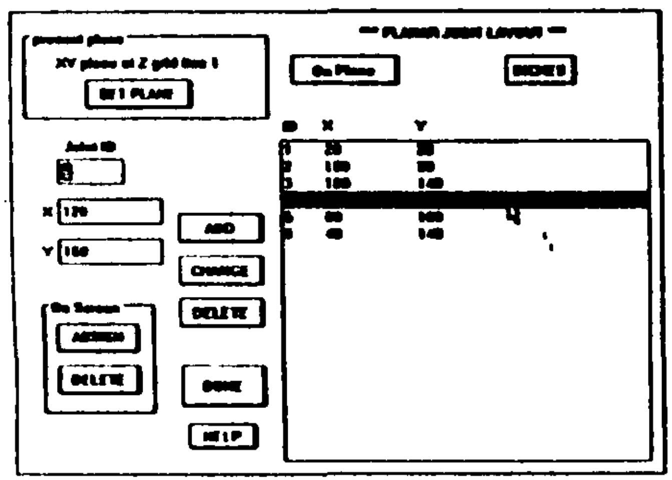
The Y grid lines are perpendicular to the Y axis and cross it at the Y coordinate specified for each line. All operations on the Y grid are exactly the same as for X grid which is described earlier.

Z Grid

The Z grid lines are perpendicular to the Z axis and cross it at the Z coordinate specified for each line. All operations on the Z grid are exactly the same as for X grid which is described earlier.

Joints

Clicking on Joints brings up different dialog boxes depending on whether 2D or 3D is set in the Control Panel. If 2D is set then the following dialog box appears:



The Planar Joint dialog box allows you to define joints on grid planes. The grids must have been previously defined. Joints can be defined by specifying their coordinates on the plane or by using the mouse to place joints at grid intersections (described later in this section).

Joints defined on grid planes are in no way different from joints defined with 3D coordinates. The only difference is that the coordinate of the grid plane is used for the 3rd coordinate. The definition of joints with 3D coordinates is described later in this section.

The first thing to do is to make sure that you are defining joints on the correct grid plane. The present plane shows the plane on which joints will be defined. If you want to change this, click on the SET PLANE push button to select the correct plane. This brings up the PLANE SELECT dialog box, which is also brought up by clicking on the plane push button in the Control Panel. Refer to the 2D and 3D Display section at the beginning of this chapter for the use of the PLANE SELECT dialog box.

List of Joints

The button above the PLANAR JOINT LAYOUT list box allows you to see the list of joints in various ways. ALL shows all existing joints, whether or not they are on a grid plane. The joints on the present plane have an asterisk to mark them. **On Plane** shows only the joints on the present plane. **Packed List** is a special combined list of joints and generations in the order in which they were defined. The Packed List is used when you do **Save** or **Save as**. You can NOT add or delete directly from the Packed List; it is for reference only.

Assigning Joints

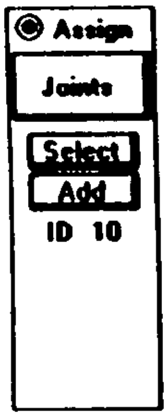
With the correct plane selected, you can define joints either Off Screen or On Screen. Note that the ID is just an identifying number and does not have to be in any order. You can NOT have two joints with the same ID, but you can have two or more joints at the same location as long as they have different IDs.

Off Screen

Enter the desired coordinates in the edit boxes, enter the ID number and click on ADD to add a new joint.

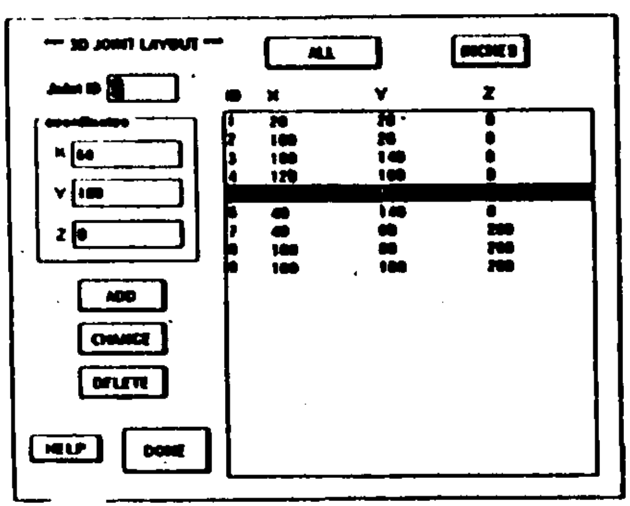
Click on an existing joint in the list box to bring its coordinates into the edit boxes. Note that it must be on the present plane. You can then change the coordinates and click on CHANGE to change it or click on DELETE to delete it.

On Screen
Specify an ID number and click on ASSIGN in the On Screen group to display the present plane with its grid lines. Click on any grid line intersection to assign a new joint. The Joint ID will be automatically set to the next higher available number so you can keep assigning joints. If you want to use other values of Joint ID, click on the Select button in the Control Panel which is shown at the left.



Click on DELETE in the On Screen group to display the present plane with its grid lines and existing joints. Click on any existing joints to DELETE them.

3D Joints:
Joints can be defined anywhere in 3 Dimensional space, by specifying the X, Y and Z coordinates. When you are in 3D, Clicking on **Layout** menu, **Joints** shows this dialog box:



The Joint ID is just an identifying number for the joint and does not have to be in any order. However, you can not have two joints with the same joint ID. See the Planar Joints dialog box for a description of the ALL and Packed Lists.

Off Screen
Enter the desired coordinates in the edit boxes, enter the joint ID and click on ADD. Click on an existing joint in the list box to bring its coordinates into the edit boxes. You can then change the coordinates and click on CHANGE to change it or click on DELETE to delete it.

On Screen
You can NOT assign joints On Screen in 3D display, but only in the 2D display.

Joint Generation

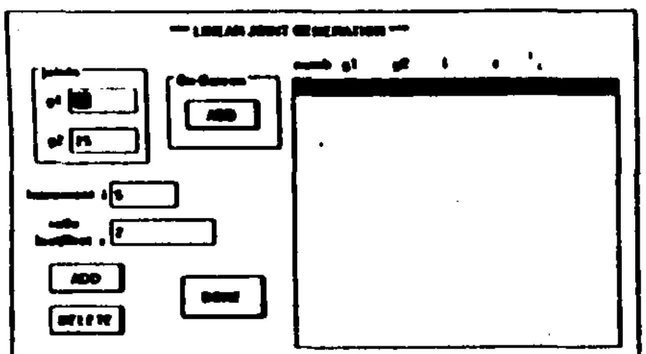
By defining a small number of joints and using **Joint Generation**, you can generate arrays of joints in the form of lines and areas. There is a complete description of this process in the SAP90 Users Manual "Input Data - JOINTS Data Block", including examples for each type of generation. PLEASE read that section and refer to it when attempting to generate joints. Following is a description of how to use the generations.

Generate Help

Clicking on **Generate Help** brings up a dialog box with abbreviated assistance for joint generation procedures.

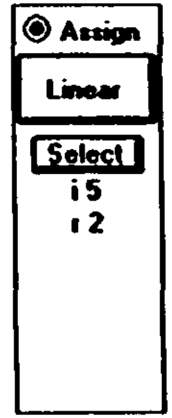
Linear

Clicking on **Linear** brings up this dialog box:



Linear generation generates a number of joints that lie on a straight line between two existing joints. The generated joints can be evenly spaced or spaced at a proportionally increasing or decreasing rate.

The numb refers to the identification number of the generation, and is NOT a joint identification number. The generation numb starts with 1 and automatically increases by one for each generation added. Generation can be done either Off Screen or On Screen as desired.



On Screen

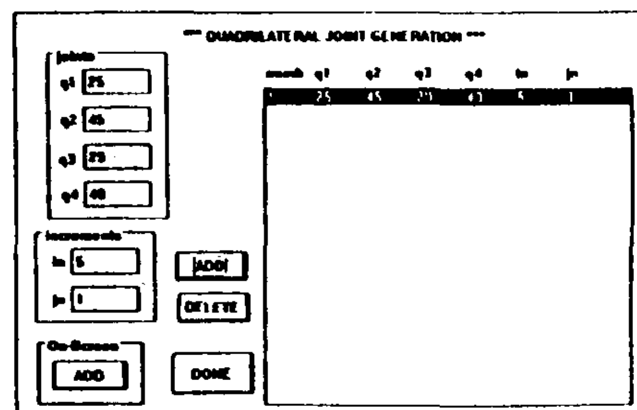
Enter the values for increment and ratio in the edit boxes and then click on ADD. When the display appears, click on the first existing joint and then click on the second existing joint. To determine the correct joints before doing the generation, RIGHT click on existing joints to display their number. This does not have any effect on the generation.

Off Screen

Enter the existing joint identification numbers in the g1 and g2 edit boxes. Enter the joint number increment between generated joints and the space ratio (1 is evenly spaced) in the edit boxes. Click on ADD to add the generation. Clicking on an existing generation in the list box will bring its values into the edit boxes. You can then delete the generation by clicking on DELETE.

Quadrilateral

Clicking on **Quadrilateral** brings up this dialog box:



Quadrilateral generation generates a two dimensional array of joints from four existing joints at the corners. The order of specifying the existing corner joints is important, either

Off Screen or On Screen. Refer to the SAP90 Users Manual "Input Data - JOINTS Data Block" for a diagram. If you end up with an unexpected looking generation, probably the order is wrong.

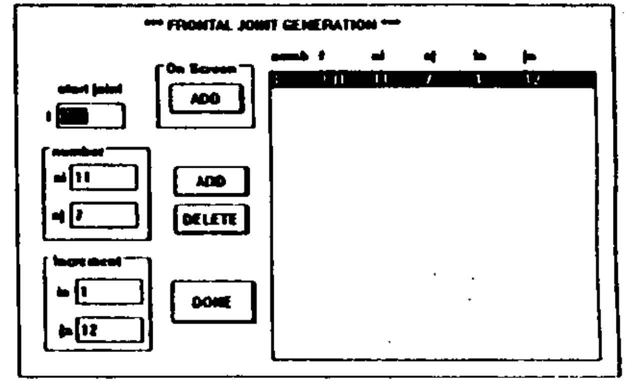
The numb refers to the identification number of the generation, and is NOT a joint identification number. The generation numb starts with 1 and automatically increases by one for each generation added. Generation can be done either Off Screen or On Screen as desired.

On Screen
Enter the values for joint number increment on each axis into the edit boxes and then click on ADD. When the display appears, click on the four existing joints in the correct order. To determine the correct joints before doing the generation, RIGHT click on existing joints to display their number. This does not have any effect on the generation.

Off Screen
Enter the existing joint identification numbers in the q1, q2, q3 and q4 edit boxes. Enter the joint number increment between generated joints on each axis in the edit boxes. Click on ADD to add the generation. Clicking on an existing generation in the list box will bring its values into the edit boxes. You can then delete the generation by clicking on DELETE.

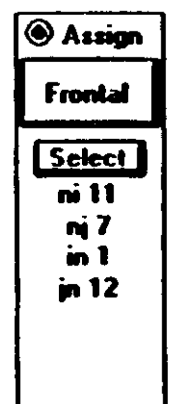
Frontal

Clicking on **Frontal** brings up this dialog box:



Frontal generation generates an array of joints in a rectangular or parallelogram shape, using existing joints along two sides of the parallelogram. Refer to the SAP90 Users Manual "Input Data - JOINTS Data Block" for a diagram showing joint numbering and the originating joint, f, used to define the generation.

The numb refers to the identification number of the generation, and is NOT a joint identification number. The generation numb starts with 1 and automatically increases by one for each generation added. Generation can be done either Off Screen or On Screen as desired.

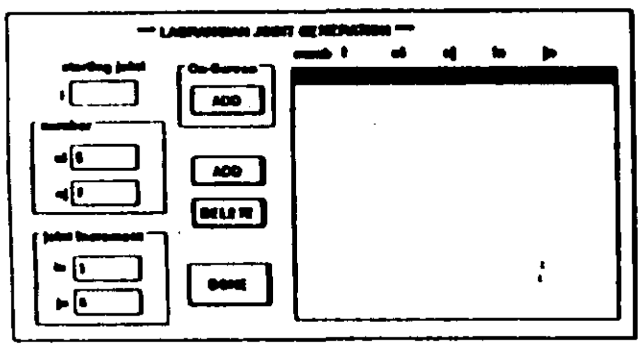


On Screen
Enter the values for the number of joints and joint number increment on each axis into the edit boxes and then click on ADD. When the display appears, click on the existing originating joint. To determine the correct joint before doing the generation, RIGHT click on existing joints to display their number. This does not have any effect on the generation.

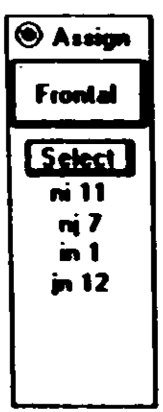
Off Screen
Enter the existing originating joint identification number f in the edit box. Enter the number of joints and joint number increment for each axis in the edit boxes. Click on ADD to add the generation. Clicking on an existing generation in the list box will bring its values into the edit boxes. You can then delete the generation by clicking on DELETE.

Lagrangian

Clicking on **Lagrangian** brings up this dialog box:



Lagrangian generation generates a four-sided array of joints that can have an arbitrary shape. ALL of the joints along the four sides must exist and be numbered correctly. Refer to the SAP90 Users Manual "Input Data - JOINTS Block" for a diagram showing joint numbering and the joint used to define the originating joint number.



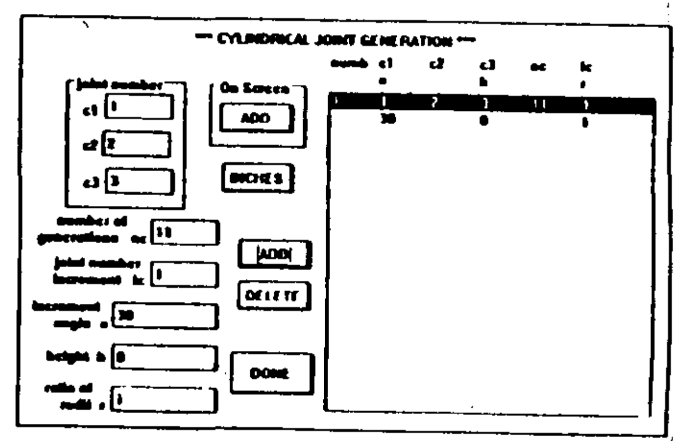
The numb refers to the identification number of the generation, and is NOT a joint identification number. The generation numb starts with 1 and automatically increases by one for each generation added. Generation can be done either Off Screen or On Screen as desired.

On Screen
Enter the values for the number of joints and joint number increment on each axis in the edit boxes and then click on ADD. When the display appears, click on the existing originating joint. To determine the correct joint before doing the generation, RIGHT click on existing joints to display their number. This does not have any effect on the generation.

Off Screen
Enter the existing originating joint identification number I in the edit box. Enter the number of joints and joint number increment for each axis in the edit boxes. Click on ADD to add the generation. Clicking on an existing generation in the list box will bring its values into the edit boxes. You can then delete the generation by clicking on DELETE.

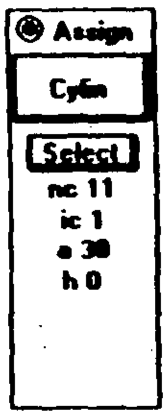
Cylindrical

Clicking on **Cylindrical** brings up this dialog box:



Cylindrical generation generates a circular array of joints around the axis of a cylinder. The array can be in a plane normal to the axis, or can be a spiral array on the surface of a cylinder or a cone. A diagram of this generation showing all the parameters is in the SAP90 Users Manual "Input Data - JOINTS Data Block".

The numb refers to the identification number of the generation, and is NOT a joint identification number. The generation numb starts with 1 and automatically increases by one for each generation added. Generation can be done either Off Screen or On Screen as desired.

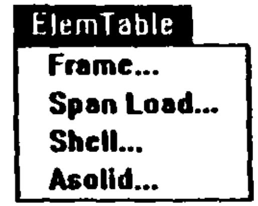


On Screen

Enter the values for the parameters (except for c1, c2 and c3) into the edit boxes and then click on ADD. When the display appears, click on c1, c2 and c3 in that order. To determine the correct joint before doing the generation, RIGHT click on existing joints to display their number. This does not have any effect on the generation.

Off Screen

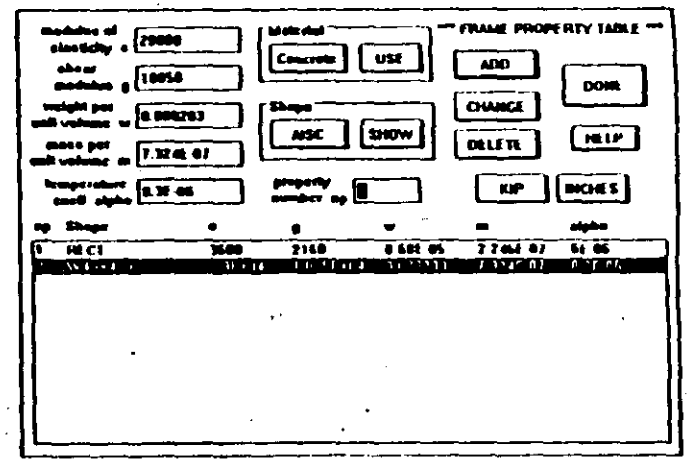
Enter the existing joint identification numbers c1, c2 and c3 in the edit boxes. Enter the other parameters in the edit boxes. Click on ADD to add the generation. Clicking on an existing generation in the list box will bring its values into the edit boxes. You can then delete the generation by clicking on DELETE.



The **ElemTable** menu allows you to define the materials and the properties of the structural elements - frames, shells and asolids. You can also define span loading patterns which can be applied to frame elements in the completed structure. Following is a definition of the **ElemTable** commands:

Frame

When you click on **Frame**, this dialog box appears:



The purpose here is to define a set of structural element properties that can later be assigned to the structure. For each element, you must specify a material and a shape. For example, one element could be a steel W40X244 and another could be a concrete pipe.

You can specify the material by entering e, g, w, m and alpha in the edit boxes, or alternatively, you can select a material using the Material push button and then clicking on USE, which fills in values for e, g, w, m and alpha. The values used in the Material box can be specified in the

Preferences menu so you can use your own values. Refer to the **Preferences** section later in this chapter.

Specify a shape by clicking on the Shape push button which will cycle through a set of available shapes. There are a number of shapes, including AISC sections, geometrical shapes and a general USER shape. Each of these shapes has a different set of parameters and some have more than others. Clicking on SHOW will bring up a dialog box which differs for the different shapes and is described on the next page.

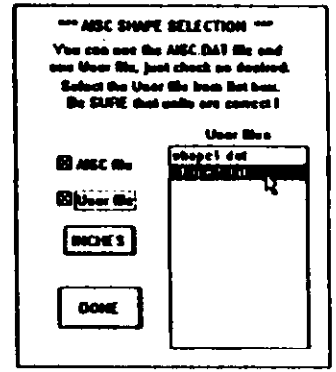
With a material and shape specified, enter the element property identification number np, which must be consecutive from one.

Finally, click on ADD to add a new Frame element property. If np already exists in the list of Frame elements, the new element will be inserted ahead of the existing element and the existing Frame elements will be renumbered.

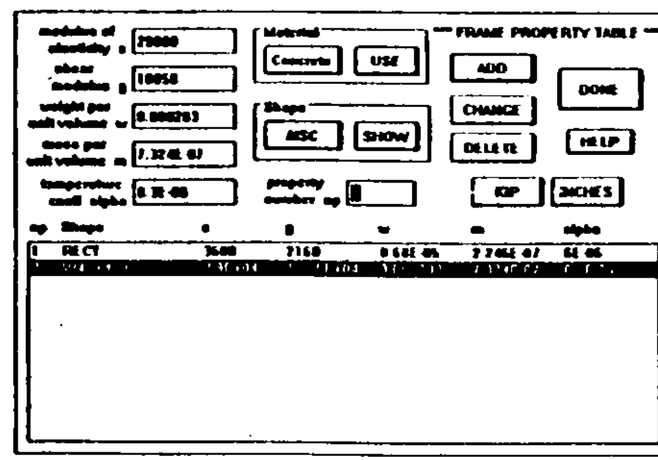
Clicking on an existing Frame element in the list box will bring its parameters up into the edit boxes. You can then change it by changing the parameters and clicking on CHANGE . Clicking on DELETE will delete the existing Frame element.

Frame Shapes

The AISC shape requires further selection from the AISC section table. When AISC is selected for the first time, clicking on the SHOW button brings up the dialog box at left. Note that this will only be done once, after that, the dialog box below will appear instead.

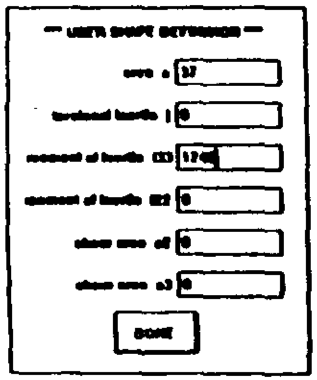


If files with a .DAT extension (other than AISC.DAT) are present in the directory where SAPIN.EXE resides, they will be shown in the list box. You can use the AISC.DAT file and ONE user file if they exist. It is VERY important that the units push button match the units in the AISC.DAT file and the user file. If they do not, then the values for weight (w) and mass (m) will be incorrect. Check the check boxes as desired and then click on DONE, which brings up this dialog box.



With AISC in the shape box, click on SHOW to select a section. The AISC sections are divided into groups for ease of selection. Click on the group desired and then select the desired section by clicking on it in the list box. Finally, click on DONE and return to the FRAME PROPERTY TABLE dialog box. The AISC section that you selected will be used whenever you do an ADD or CHANGE.

Geometrical and User shapes all have additional parameters, but the number of parameters vary. All these shapes require you to enter the dimensions of the shape and work in the same way, so an example of USER is shown at the left. If USER is selected, click on SHOW to bring up this dialog box.

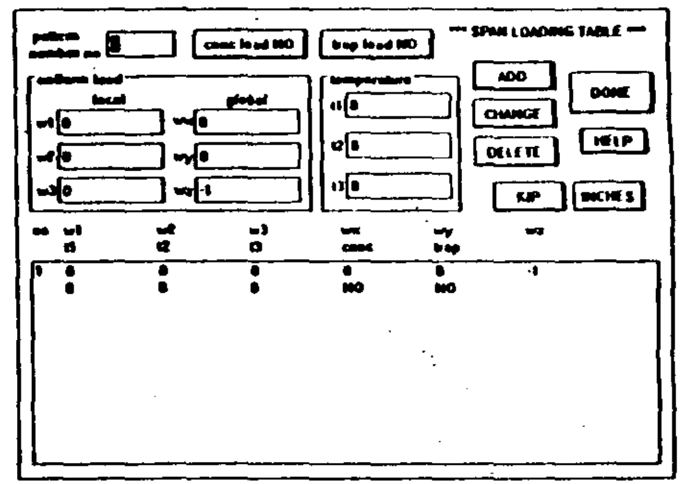


Enter the parameters as desired and click on DONE. The geometrical shapes all have similar parameters but may have fewer than six.

Span Load

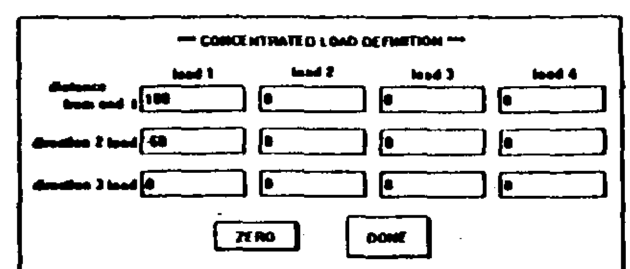
Span Load patterns are treated in the same way as Frame section property types, that is, a set of Span Load types is defined. When the Span Loads are later assigned to Frame elements, they are picked from the list of Span Loads that you have defined.

Clicking on **Span Load** brings up this dialog box:



For uniform span loads, enter the parameters w1,w2,w3,wx,wy,wz and t1,t2,t3 as desired. Note that ns is the Span Load identification number, and must be consecutive from one.

If you need to enter or view concentrated loads, click on conc load to bring up this dialog box:



If you have ANY concentrated loads defined, the button will show conc load YES. Otherwise, it will show conc load NO.

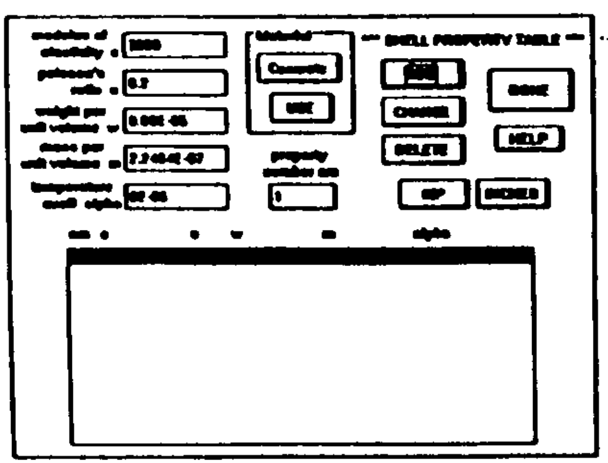
A similar dialog box is used for trapezoidal loads. Please refer to the SAP90 Users Manual "Input Data - FRAME Data Block" for a description of how these loads are defined. If distances are positive, then they must be increasing. If distances are negative, then they are ratios, must be between 0.0 and -1.0, and must be increasingly negative.

In the same way as for concentrated loads, the trap load buttons will say NO if there are no loads defined and YES if there are.

After defining any trapezoidal or concentrated loads, make sure that ns is set correctly and use the ADD, CHANGE and DELETE BUTTONS to make the list of Span Loads in the same way as for Frame section properties.

Shell

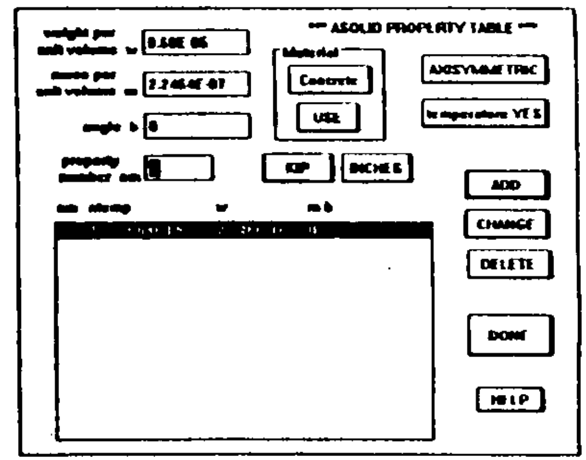
Shell material properties are defined like Frame section properties. Clicking on **Shell** brings up the following dialog box:



Enter values for the parameters as desired; then enter nm and use the ADD, CHANGE and DELETE BUTTONS to make the list of Shell material properties in the same way as for Frame section properties. Note that nm is the Shell material property type identification number and must be consecutive from one.

Asolid

Asolid material properties are defined in much the same way as Frame section properties. Clicking on **Asolid** brings up this dialog box:

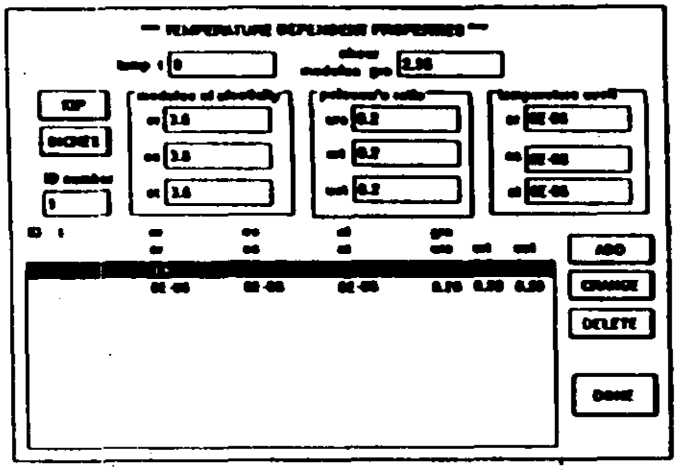


Enter the parameters w, m, b and the Asolid material property type identification number, nm. Note that nm must be consecutive from one.

Click on the push button to select one from Axisymmetric, Plane Strain or Plane Stress.
CAUTION; There can ONLY be ONE selection that applies to ALL Asolid elements, so that the selection existing when the file is SAVED is the setting written to the file. In other words, set this once and don't change it.

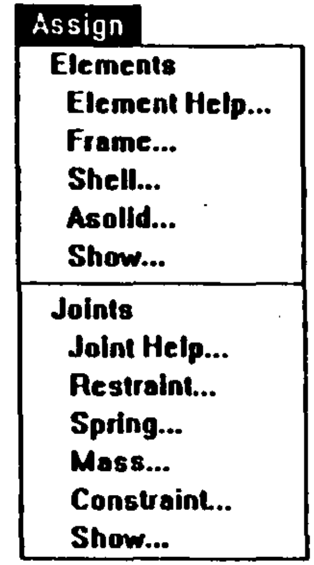
You will need some temperature dependent properties. Note that the temperature button says NO if there are no temperature dependent properties and YES if there are.

Click on temperature to bring up the following dialog box:



Enter the temperature dependent parameters and the ID, which must be consecutive from one. Click on ADD, CHANGE or DELETE to build up a list of temperature dependent properties for the current Asolid material property type. Click on DONE to return to the Asolid dialog box.

Finally, use the ADD, CHANGE and DELETE buttons in the Asolid dialog box to make a list of Asolid material property types, again using the same procedure as for Frame section property types.



The **Assign** menu allows you to assign structural members such as Frames and Shells to the current structure. You can also assign various items to joints, such as restraints and masses. Before starting **Assign**, all joints should be defined, if at all possible. It is MUCH better to have all joints defined before assignment, than to change the joint layout afterwards. Changing the joint layout after assignments has begun may cause existing assignments to be deleted or moved, and you will then have to redo the assignments. Please refer to the SAP90 Users Manual "Input Data" under the section for the type of assignment.

Elements

All structural elements are assigned between existing joints, using the joints to define the boundaries of the element. Regardless of the type of element, you specify the desired properties for the element and then assign as many of that elements as you want. If you want to assign an element with different properties, you must stop assigning, change the properties as desired and then start assigning again.

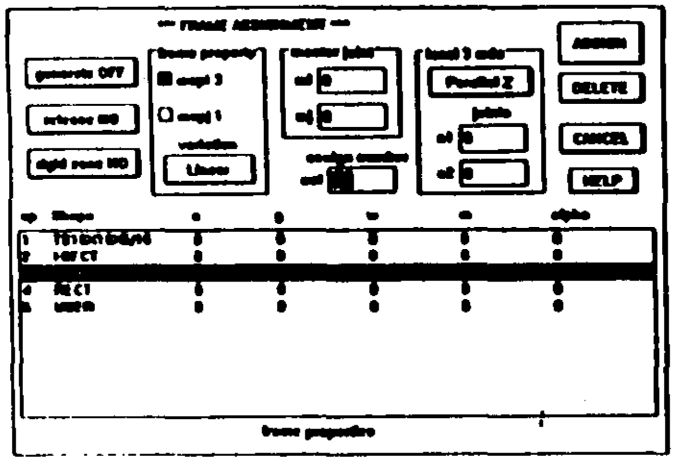
Element Help

Clicking on **Element Help** brings up a dialog box with abbreviated information about assigning elements.

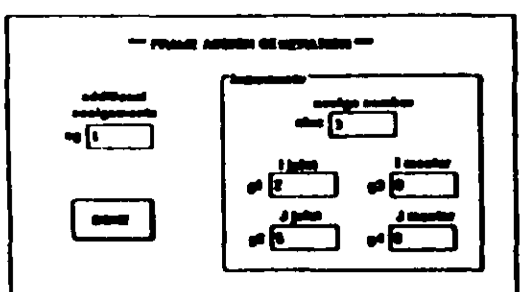
Frame

In addition to the Frame element section properties defined in the **ElemTable** menu, Frame elements have a large number of other items such as end condition t must be specified. Keep in mind that when you are assigning Frames, each assignment has the properties and conditions that are presently set in the various dialog boxes reached through the frame assignment dialog box.

Clicking on **Frame** brings up the following dialog box:

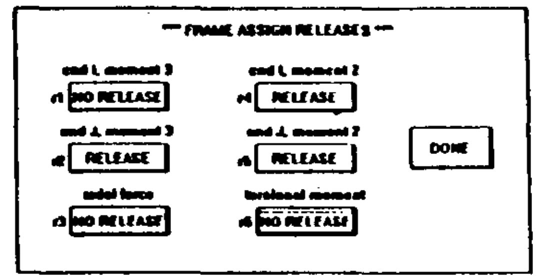


You MUST refer to the SAP90 Users Manual "Input Data - FRAME Data Block" for the definition of the items used in this dialog box. The 3 push buttons at the left bring up additional dialog boxes to specify releases, rigid zones and generations. If any of these items have been defined, then the button will show YES, otherwise it will show NO. Clicking on generate brings up the following dialog box:



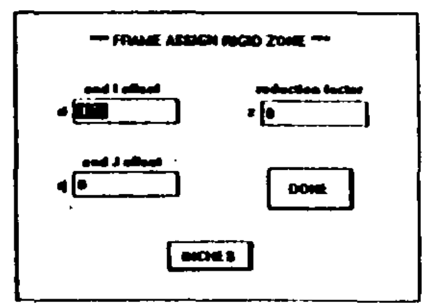
You can specify additional Frame elements to be automatically generated when you assign a single Frame, by entering ng larger than zero and entering the other parameters as required. The generated frames use existing joints and the numbering of the joints must be correct. Please refer to the SAP90 manual for the exact definitions for the parameters.

Clicking on release brings up this dialog box:



Click on the push buttons r1 through r6 to set the releases as desired and then click on DONE.

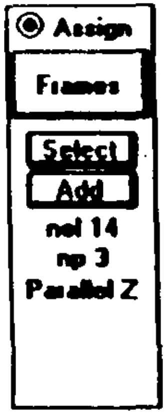
Clicking on rigid zone brings up this dialog box:



Enter the offsets and reduction factor as desired and then click on DONE.

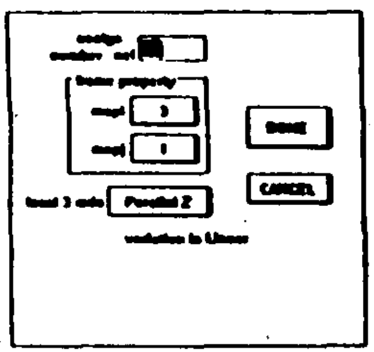
Returning to the Frame Assignment dialog box, set mspi and mspj by checking the desired button and then clicking on the desired property in the list box.

Click on the variation button to set it as desired. Click on the local 3 axis button to set it. In some cases, you may need to specify n1 and n2. Specify the master joint if desired. Finally, enter the desired assign number, nel, (element identification number in the SAP90 manual) which may have any value between 1 and 32767.



Now click on ASSIGN (or DELETE) and the display will appear. Use Limits and Zoom in the Control Panel to restrict the amount of the structure shown on screen. Too many joints or overlapping joints will make assignment difficult. As shown at left, the Control Panel will show information about the assign parameters.

Clicking on select in the Control Panel will bring up this dialog box:



You can use select to change the most used parameters. It will also show when there are generation, releases or rigid zone frame items.

To assign a Frame element, point to a joint, click the left button, move to another joint, and click on it. A Frame will be assigned, having all the properties presently set. If ng (in the generate dialog box) is grea zero, additional

Frames will be generated, assuming that the necessary joints exist. This process of assignments may be repeated as desired, but remember that all the properties and conditions remain the same.

To aid in selecting joints, RIGHT click on any joint to see its number. This will not interfere with a Frame assignment in progress.

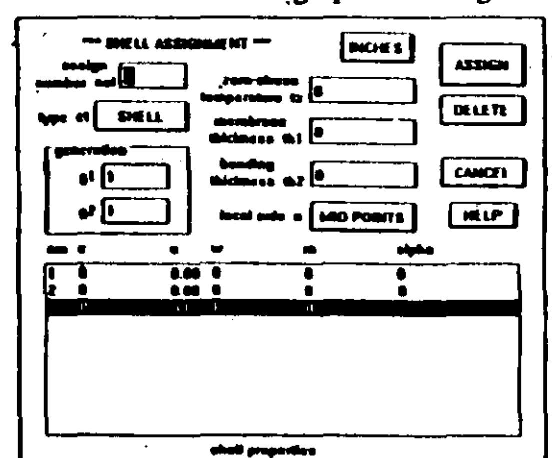
To delete a Frame element, use the same procedure as assignment but over an existing Frame.

If you have selected Frame in the **Display** menu, **Right button show**, you can use the same assignment procedure with the RIGHT button over an existing frame element to see its properties.

Frame elements are shown in RED except for those that have the same value of mspi as is presently set. Those Frames are shown in MAGENTA. This means that Frames you are presently assigning will always show in MAGENTA until you change mspi to a different value.

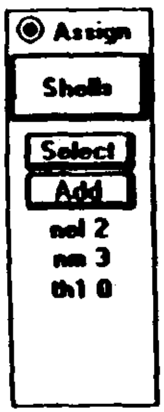
Shell

Click on Shell to bring up this dialog box:



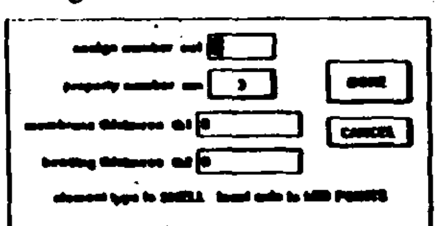
You MUST refer to the SAP90 Users Manual - "Input Data-SHELL Data Block" for definitions of the items in this dialog box. Like Frames, the properties and conditions defined in this dialog box will apply to all the Shells that you assign.

The Shell assign number, nel, (element identification number in the SAP90 manual) can be any value between 1 and 32767 and does not need to be in any order. The generation parameters g1 and g2 will generate additional Shells from the one that you assign, but the joints used for the generation must exist and must be numbered correctly.



Enter the items in the dialog box as desired and click on the desired property in the list box. Then click on ASSIGN (or DELETE) and the display will appear. Use Limits and Zoom in the Control Panel to restrict the amount of the structure shown on screen. Too many joints or overlapping joints will make assignment difficult. As shown at left, the Control Panel will show information about the assign parameters.

Clicking on select in the Control Panel will bring up this dialog box:



To assign a Shell element, click on the joints around the Shell periphery and finish by clicking on the first joint to close the Shell.

Do NOT cross over the Shell, but starting at one joint, go around the EDGE of the Shell until you return to the starting joint. Going around the shell counter clockwise points the normal to the shell out of the screen. Clockwise reverses it. You can assign Shells with either 3 or 4 joints. To aid in selecting joints, RIGHT click on any joint to see its number. This will not affect a Shell assignment in progress.

Deleting a Shell is done by using the same procedure to go around the edge of an existing Shell.

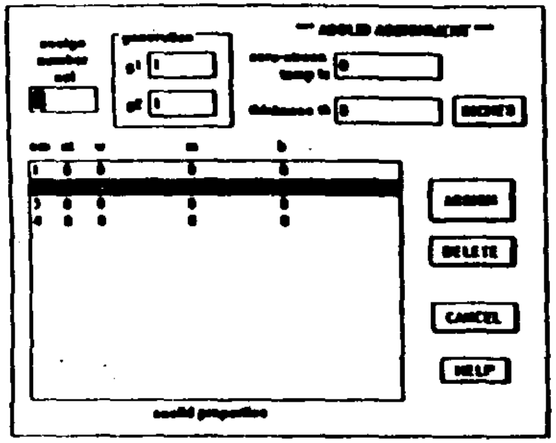
Assigned Shells will be drawn in GRAY in the 3D views except for those that have the same property as is presently selected in the Shell Assignment list box. These Shells will be shown in MAGENTA. This means that when you are assigning, the Shells will be MAGENTA until you change to another property in the list box.

Shells can be shown as filled or as outlines. This is controlled in the **Display** menu, **Display Options**.

If you have selected Shell in the **Display** menu, **Right button show**, you can use the same assignment procedure with the RIGHT button over an existing Shell to see its properties.

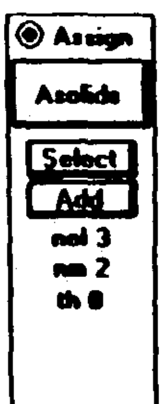
Asolid

Click on **Asolid** to bring up this dialog box:



You MUST refer to the SAP90 Users Manual - "Input Data-ASOLID Data Block" for definitions of the items in this dialog box. Like Frames, the properties and conditions defined in this dialog box will apply to all the Asolids that you assign.

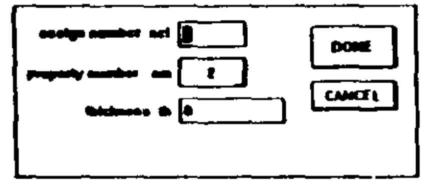
The Asolid assign number, nel, (element identification number in the SAP90 manual) can be any value between 1 and 32767 and does not need to be in any order. The generation parameters g1 and g2 will generate additional Asolids from the one that you assign, but the joints used for the generation must exist and must be numbered correctly.



Enter the items in the dialog box as desired and click on the desired property in the list box. Then click on ASSIGN (or DELETE) and the display will appear. Use Limits and Zoom in the Control Panel to restrict the amount of the structure shown on screen. Too many joints or overlappi g joints will e assig ent difficult.

As shown at left, the Control Panel will show information about the assign parameters.

Clicking on select in the Control Panel displays this dialog box:



You can assign Asolids with 4 to 8 joints and with an optional center joint. All of these Asolids have 4 corners, so that Asolids with 5 or more joints will have joints on the sides, called midside joints. You can also assign a 3 joint triangular Asolid without a center joint.

To assign an Asolid element, click on the joints around the edge of the Asolid in order, including any midside joints. Finish by clicking on the starting joint to close the Asolid. You can go around the edge either clockwise or counter-clockwise, but do NOT cross over the Asolid.

When you have clicked on the closing joint, then you will be prompted to click on the optional center joint. If you are assigning an Asolid with 5, 6 or 7 joints, you will then be prompted to click on the midside joints again to identify them. Click on them in the same order as you did the first time.

To aid in selecting joints, RIGHT click on any joint to see its number. This will not disturb the any Asolid assignment in progress.

Deleting an Asolid element is done in the same way as assigning; use the same procedure to go around the edge of an existing Asolid. You will be promp' for a center joint as in assigning, but will not need to click on midside joints again.

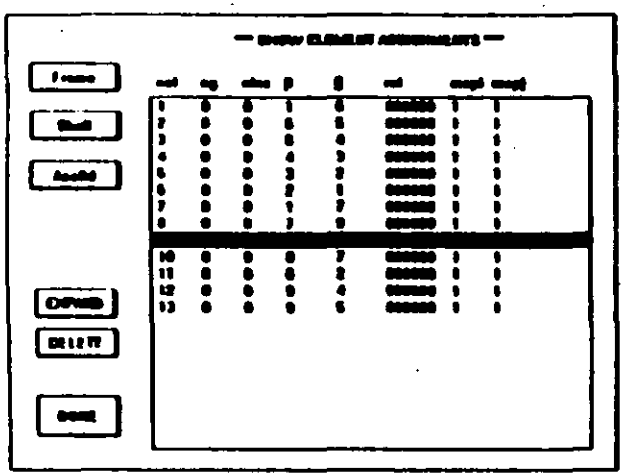
Assigned Asolids will be drawn in CYAN (light blue) except for those that have the same property as is presently selected. These Asolids will be shown in MAGENTA. This means that when you are assigning, the Asolids will be MAGENTA until you change to another Asolid property.

Asolids can be shown as filled or as outlines. This is controlled in the **Display** menu, **Display Options.**

If you have selected Asolid in **Display** menu, **Right button show**, you can use the same assignment procedure with the RIGHT button over an existing Asolid to see its properties. Like deleting, clicking on midside joints a second time is not required.

Show

Clicking on **Show** brings up this dialog box:



Click on the FRAME, SHELL or ASOLID to show all existing assignments of that type. Click on an existing assignment in the list box and then click on DELETE to delete it or click EXPAND to additi mo-

ters not shown in the list box. EXPAND only has significance for Frames.

Joints

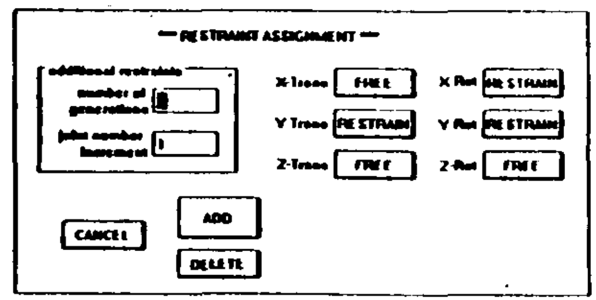
All joint assignments are assigned to existing joints. You specify the desired properties for an assignment and then can make as many assignments as desired, but all assignments have the same properties until the properties are changed.

Joint Help

Click on **Joint Help** to bring up a dialog box with abbreviated assistance about joint assignment.

Restraint

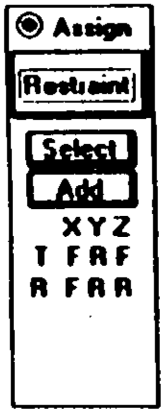
Clicking on **Restraint** brings up this dialog box:



Click on the TRANS and ROT push buttons to set the desired Restraint. Enter number of generations (ng) and joint number increment (ninc). Then click on ADD (or DELETE). When the display appears, click on a joint to assign the Restraint. If ng is greater than 0, then ng additional Restraints will be generated by successively adding ninc to the joint number.

Delete Restraints by clicking on a joint that has an existing Restraint.

You can click on Select in the Control Panel (shown at left) to bring up the same Restraint dialog box as above.

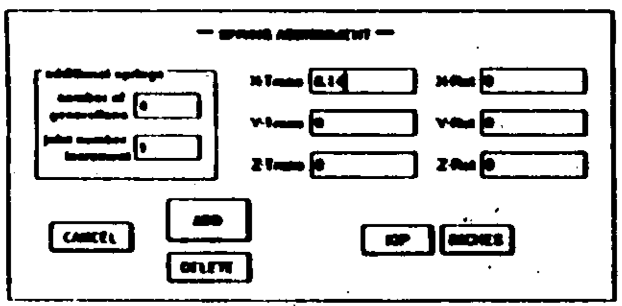


Restraints are shown as LIGHT GREEN except when the Restraint has the same properties as are presently set in the Restraint Assignment dialog box, in which case the restraint is shown as MAGENTA. This means that when you are assigning Restraints, they will be shown in MAGENTA until you change the Restraint properties.

If you select Restraint in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Restraint properties.

Spring

Clicking on **Spring** brings up the following dialog box:



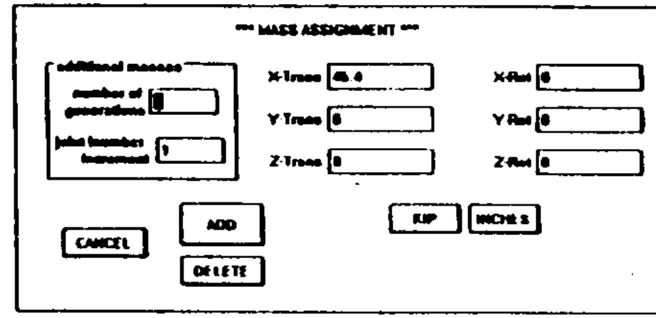
Enter the desired Spring constants in the edit boxes. Assignment or deletion is exactly the same as for Restraints, including additional Spring generations. Springs will only be shown in MAGENTA as they are assigned. Thereafter, they will be shown in LIGHT GREEN.

If you select Spring in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Spr roperties.

You can click on select in the Control Panel to bring up the same Spring dialog box as above.

Mass

Clicking on **Mass** brings up this dialog box:



Enter the desired Masses in the edit boxes. Assignment or deletion is exactly the same as for Restraints, including additional Mass generations. Masses will only be shown in MAGENTA as they are assigned. Thereafter, they will be shown in LIGHT GREEN.

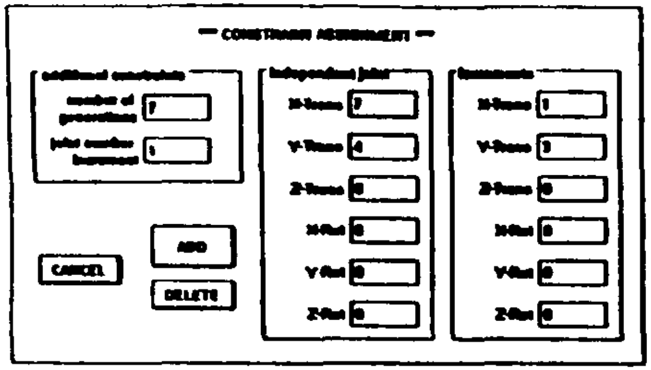
Extra care should be taken in the units being used. The units showing in the unit pushbuttons are for force and length. Mass units consistent with these should be used to enter data. For example, for the units of kip and inches showing in the dialog box above, the translational masses should be specified in units of kip-sec^2/inch and the rotational masses should be specified in the units of kip-sec^2-inch.

If you select Mass in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Mass properties.

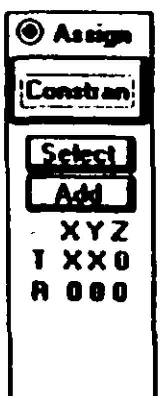
You can click on Select in the Control Panel to bring up the same Mass dialog box as above.

Constraint

Clicking on **Constraint** brings up this dialog box:



Enter the desired Constraints in the edit boxes. Assignment or deletion is exactly the same as for Restraints, including additional Constraint generations. There is the additional increment for the independent joint that is used when Constraint generation is done. That is, during generation, the independent joint number will be incremented by the values specified, in addition to the dependent joint number being incremented by joint number increment.



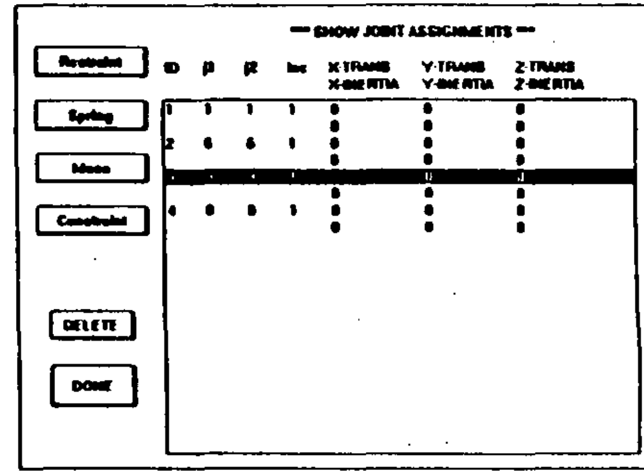
Constraints will only be shown in MAGENTA as they are assigned. Thereafter, they will be shown in LIGHT GREEN.

If you select Constraint in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Constraint properties.

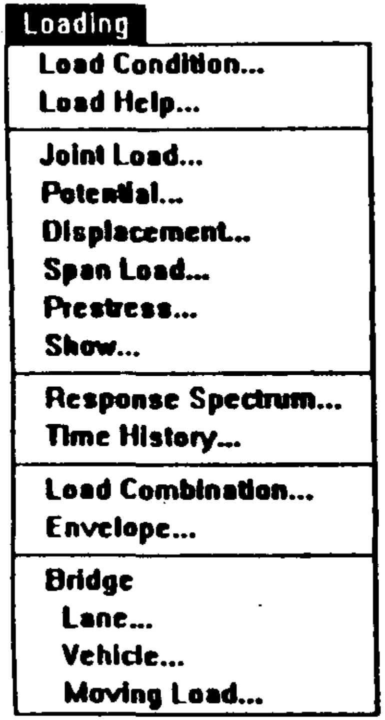
You can click on Select in the Control Panel (shown at left) to bring up the same Constraint dialog box as above.

Show

Clicking on **Show** will bring up this dialog box:



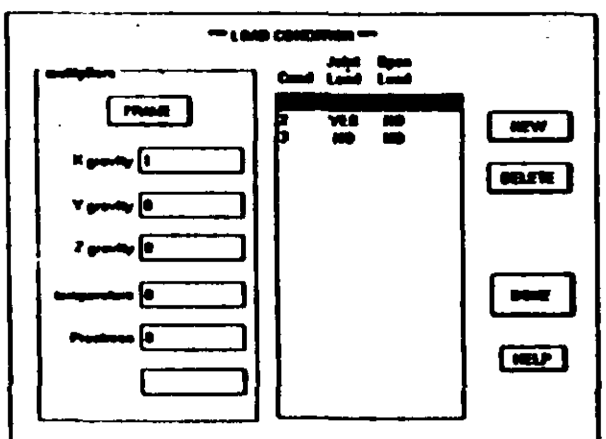
Click on RESTRAINT, SPRING, MASS or CONSTRAINT to show a list of the present assignments of that type. Click on any assignment in the list box and then click on DELETE to delete the assignment. Note that any generations will also be deleted.



The **Loading** menu allows you to define loading for your structure. Global gravitational loads can be defined. Joint loads and Frame loads can be specified, as well as joint displacements. Dynamic loading, using response spectrum and time history can also be specified. Combinations of loading conditions can be specified, using different combination multipliers for each condition. Finally, bridge lanes can be specified and various types of highway loadings applied. It is best to have the structure completely defined, with all element assignments complete, before specifying the loading.

Load Condition

Clicking on **Load Condition** brings up this dialog box:



You can specify many Load Conditions, each having its own set of multipliers, joint loads and span loads. Within each condition, there are a set of multipliers for each element type. To add a new condition, click on NEW which will add a new empty condition. Clicking on an existing condition and then clicking on DELETE will delete the existing condition and renumber all higher numbered ones.

Select a condition by clicking on it in the list box and then select the desired element by clicking on the push button in the MULTIPLIERS group. Then enter the desired multipliers. All multipliers default to zero, which means that they are disregarded. Be sure to specify the multiplier for ALL elements within each condition.

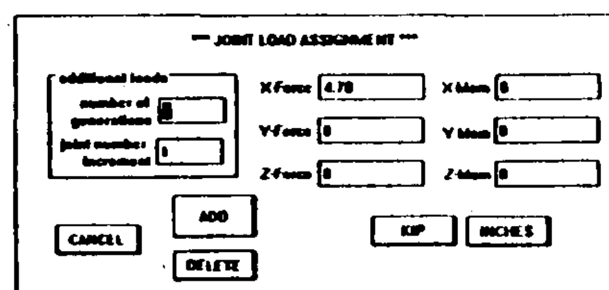
If you have defined any span or joint loads for the selected condition, then YES will show in the list box, otherwise it will show NO.

Help

Clicking on **Help** brings up a dialog box with abbreviated assistance on Load Condition and loads.

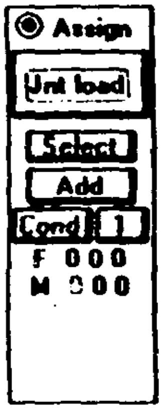
Joint Load

Clicking on **Joint Load** brings up this dialog box:



Be SURE to note that you are inputting values and assigning Joint loads for the PRESENT load condition (the one selected in the **Load Condition** dialog box). There is a different set of Joint Loads for EACH Load Condition.

Enter the desired values for Force and Moment. Enter number of generations (ng) and joint number increment (ninc). Then click on ADD to assign Joint Loads. When the display appears, click on a joint to assign the Joint Load. If



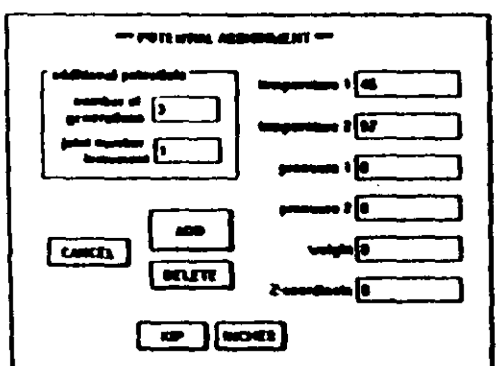
ng is greater than 0, then ng additional Joint Loads will be generated by successively adding ninc to the joint number. Of course, the joints must exist.

Delete Joint Loads by clicking on a joint that has an existing Joint Load.

Joint Loads are shown in MAGENTA as they are assigned. Thereafter, they will be shown in LIGHT GREEN.

You can click on Select in the Control Panel (shown at left) to bring up the same Joint Load dialog box as above.

If you select Joint Load in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Joint Loads.



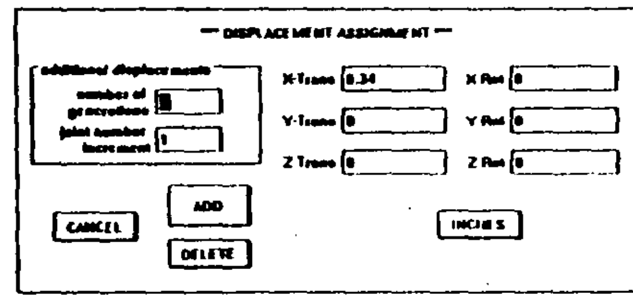
Potential

Clicking on **Potential** brings up this dialog box:

There can only be one Potential assigned to a joint, regardless of the number of Load Conditions. Otherwise, Potentials are assigned and deleted exactly the same as Joint Loads.

If you select Potential in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Potential properties.

You can click on Select in the Control Panel to bring up the same Potential dialog box as above.



Displacement

Clicking on **Displacement** shows this dialog box:

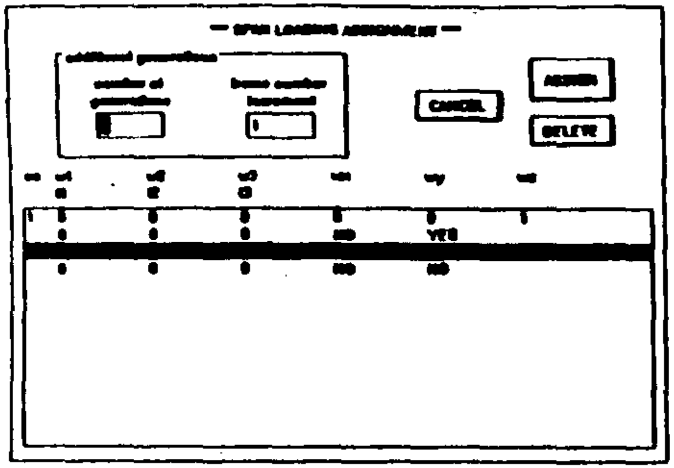
There can only be one Displacement assigned to a joint, regardless of the number of Load Conditions. Otherwise, Displacements are assigned and deleted exactly the same as Joint Loads.

If you select Displacement in the **Display** menu, **Right button show,** you can RIGHT click on a joint to determine its Displacement properties.

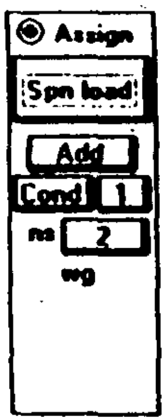
You can click on select in the Control Panel to bring up the same Displacement dialog box as above.

Span Load

Clicking on **Span Load** brings up this dialog box:



Be SURE to note that you are assigning Span loads for the PRESENT load condition (the one shown in the Control Panel at left). There is a different set of Span Loads for EACH Load Condition.



Select the desired Span Load from the list box. Enter number of generations (ng) and frame number increment (ninc). Then click on ADD to assign Span Loads. When the display appears, click on one end of an existing Frame element and then click on the other end. If ng is greater than 0, then ng additional Span Loads will be generated by successively adding ninc to the Frame element number. (The Frames must exist.)

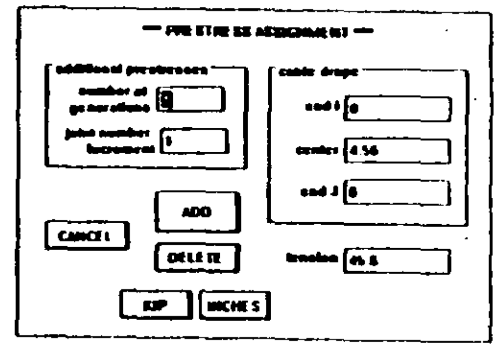
Delete Span Loads in the same way as Assigning. You can NOT delete a generated Span Load, but must delete the original Span Load that caused the generation, which deletes the original Span Load and all of its generations.

Span Loads are shown in BLUE, except when the Span Load is the same as the one currently selected in the Span Loading Assignment list box, in which case it is MAGENTA. This means that when you are assigning Span Loads, that they will be shown as MAGENTA until you change the Span Load in the list box.

If you select Span Load in the **Display** menu, **Right button show,** you can use the same procedure as assigning (but with the RIGHT button) to determine its Span Loads.

Prestress

Click on **Prestress** to show this dialog box:



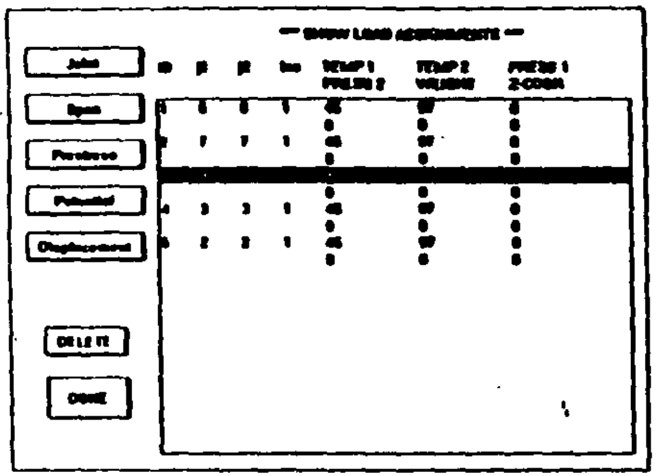
Prestress applies to ALL Load Conditions, that is, there is only one Prestress pattern for each Frame element regardless of the number of Load Conditions. The Prestress load applied to a particular Load Condition consists of the Prestress pattern load modified by the Prestress multipliers which are defined in the Load Condition dialog box.

Enter the cable drape, tension, number of generations and joint number increment. Then assign and delete Prestress in exactly the same way as Span Loads.

You can click on select in the Control Panel to bring up the same Prestress dialog box as above.

Show

Clicking on **Show** brings up the following dialog box:

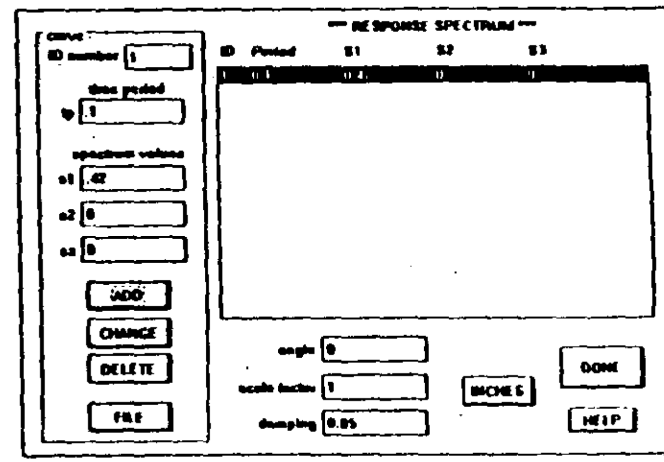


Clicking on any of the load type push buttons will show the assignments for that type. Clicking on an existing assignment in the list box and then clicking on DELETE will delete that assignment.

CAUTION:
the list of Joint Loads and Span Loads are only for the PRESENT Load Condition, that is, the Load Condition that is selected in the **Load Condition** dialog box, while there is only ONE set of Potential, Displacement and Prestress for ALL Load Conditions.

Response Spectrum

Before starting Response Spectrum, make sure that nfq or nritz is set to non-zero in the **System** menu, **System Block** dialog box. Otherwise, Response Spectrum will be disregarded.



Clicking on **Response Spectrum** shows this dialog box:

The purpose here is to define a curve for spectrum analysis. Refer to the SAP90 Users Manual "Input Data - SPEC Data Block" for a complete description of the Response Spectrum definition and use.

You can enter the curve point by point by entering ID, tp, s1, s2 and sz and clicking on ADD to add (insert) a new point in the curve. The ID is simply an identification number for the curve points and must be consecutive from one. The time period, tp, must be positive and increasing.

Clicking on an existing point in the list box will bring its values into the edit boxes. They can then be changed and clicking on CHANGE will change the point. Clicking on DELETE will delete the point.

The curve can also be read in from a file. The file must be an ASCII file with one line for each point. You can NOT use arithmetic calculations in the file. Each point must have tp, s1, s2, and sz in that order, separated by commas or spaces. The only valid characters are: 0 1 2 3 4 5 6 7 8 9 - + e E

An example of a valid file is:

0	0.30	3e-1	.20
0.1	0.35	0.35	0.23
.2	0.7	7E-1	0.45

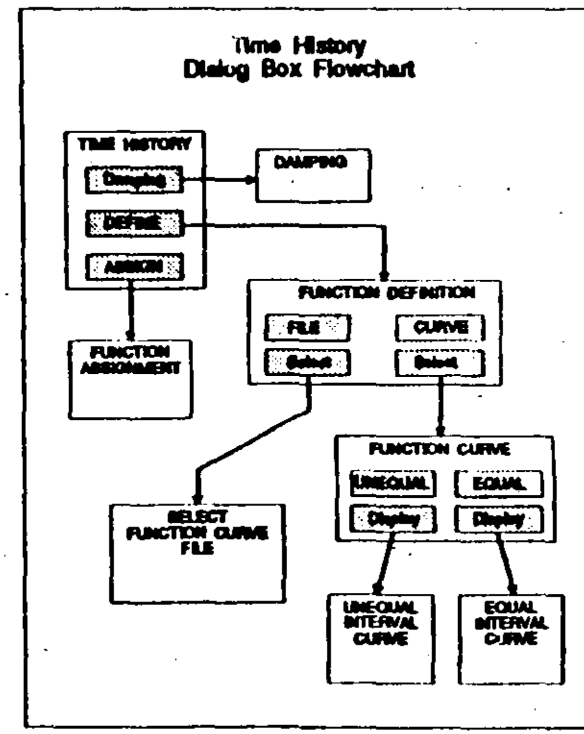
Click on FILE and then select a file using the same procedure as for **Open** in the **File** menu. A curve read in from a file can be edited using ADD, CHANGE and DELETE as described above.

Be sure to set Angle, Scale Factor and Damping before clicking on DONE.

Time History

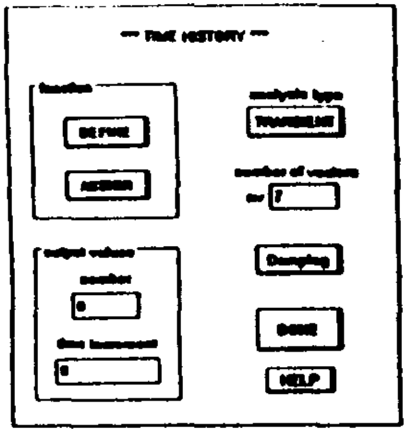
Before starting Time History, be sure to set nritz or nfq (in the **System** menu, **System Block** dialog box) to non-zero, otherwise the Time History data will be disregarded.

Refer to the SAP90 Users Manual "Input Data - TIMEH Data Block" for a complete description of Time History procedures. Time History has several dialog boxes associated with it as shown below.

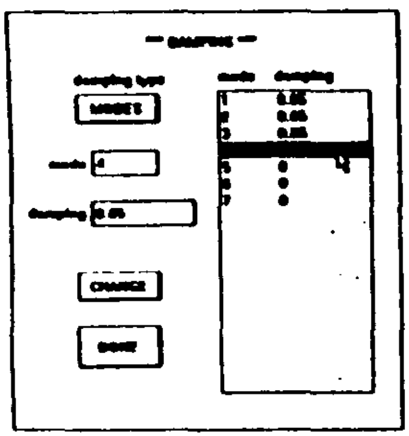


The process is to define a number of Time History functions and then to assign the functions. The functions are curves, which can exist in a file or the data points for the curves can be entered using the Equal or Unequal Interval Curve dial boxes.

Clicking on **Time History** brings up this dialog box:



Enter the number of vectors, which must be less than or equal to nritz or nfq. Click on Damping to bring up this dialog box:

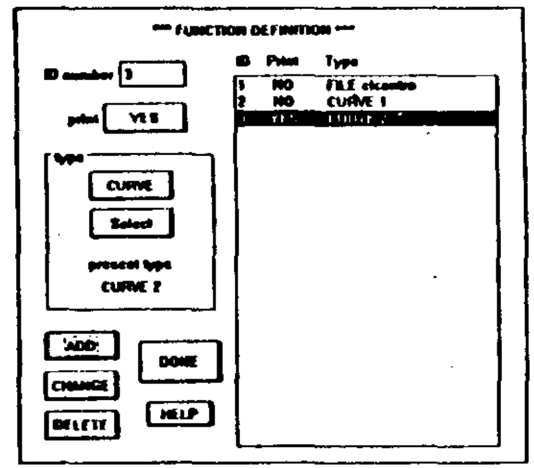


Click on the damping type push button to select CONSTANT or MODES. If you select CONSTANT, then enter the damping value. If you select MODES, then you will see a list of values. Click on a value in the list box, change the

value in the edit box, and click on CHANGE to change the value in the list. Finally, click on DONE.

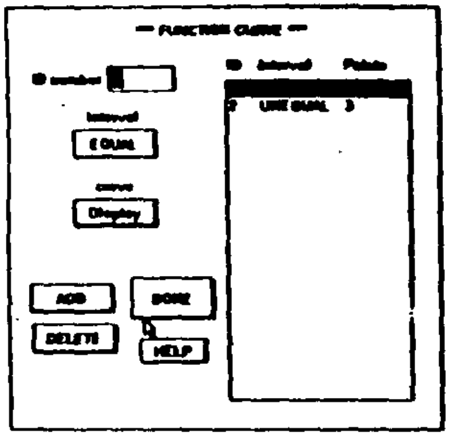
Now that damping has been defined, you need to define a set of Functions and assign them to the various Load Conditions. Therefore, before continuing, be SURE that you have defined the Load Conditions in the **Load Condition** dialog box.

Click on DEFINE to bring up this dialog box:



There are two types of Functions that can be defined, FILE and CURVE. They both define a Time History curve, but CURVE allows you to enter data to define the curve, while FILE is the name of a file which contains the curve. The file must be in the format specified in the SAP90 Users Manual "Input Data - TIMEH Data Block". The data in the file is not read or checked in SAPIN. The file must be available in the same directory as the SAP90 data file, when you run SAP90, which will read and check it. Data entered in the CURVE Function is checked as it is entered.

If you have CURVE Functions, it is best to finish defining the curves before putting them into the Function list. To define curves, click on the top push button in the TYPE group until CURVE is shown and then click on SELECT to bring up the following dialog box:

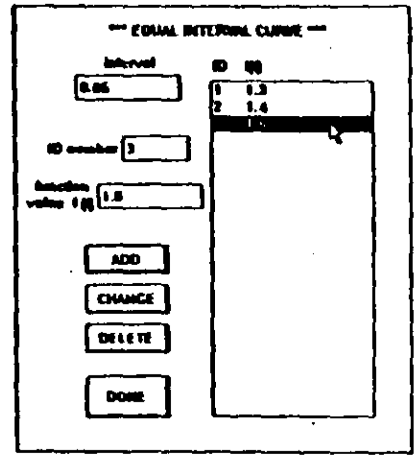


You can have curves with either EQUAL or UNEQUAL time spacing. In order to define the points in a curve, it is first necessary to have the curve in the list box. Click on the interval push button to set EQUAL or UNEQUAL, enter the desired curve ID and click on ADD. The curve ID must be consecutive from one and is used to identify the curve.

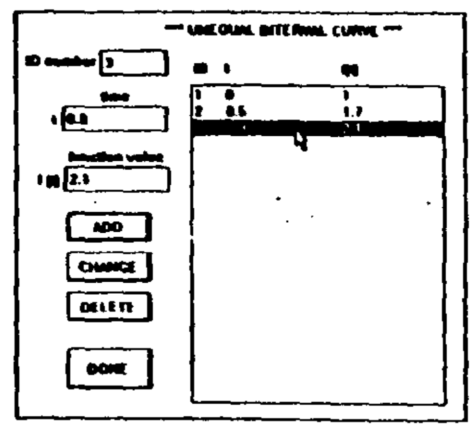
You can click on an existing curve in the list box to bring its interval and curve ID into the edit boxes. Then click on DELETE to delete the curve.

Once a curve has been put into the list box, you can now define the points in the curve.

To define the points contained in the curve, click on an existing curve in the list box and then click on DISPLAY. For EQUAL curves, this dialog box will appear:



For UNEQUAL curves, this dialog box will appear:



The UNEQUAL curve requires a time value for each point, which must be positive and increasing. The EQUAL curve requires an interval value, which is the time between points.

In either case, to add a point, enter the desired data in the edit boxes and click on ADD. The point ID must be consecutive from one, and is just the point number.

Clicking on an existing point in the list box will bring its values into the edit boxes. Change the values and click on CHANGE to change the existing value. Click on DELETE to delete the existing value.

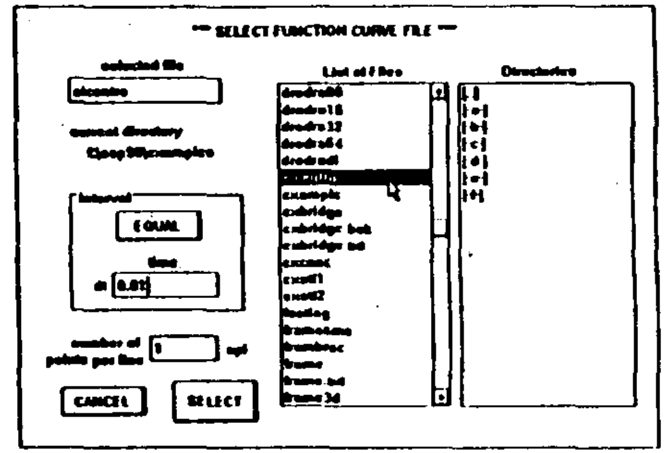
When the curve points have been defined, click on DONE to return to the dialog box with the list of curves (Function Curve dialog box). Repeat the same procedure to define the points for all of the curves in the list box. When complete, you will have a list of curves that you can assign to Functions. Finally click on DONE in the Function Curve dialog box.

You are now in the Function Definition dialog box, and ready to make a list of Functions. Each Function has an ID, which must be consecutive from one and is the Function number. You can also specify PRINT for each Function.

To add a Function using a CURVE, set the TYPE to CURVE and click on the SELECT button. This will bring up the Function Curve dialog box. Click on one of your curves in the list box and then click on DONE. You will see that the present type is the curve you selected. Click on ADD to add the curve to the list.

Click on an existing Function in the list box to bring its values into the edit boxes. You can then change the values and TYPE as desired and click on CHANGE to change the Function in the list box. Clicking on DELETE will delete the Function.

So far, you have only used CURVE Functions, but it is also possible to define FILE Functions. Set the TYPE to FILE and then click on SELECT to bring up this dialog box:



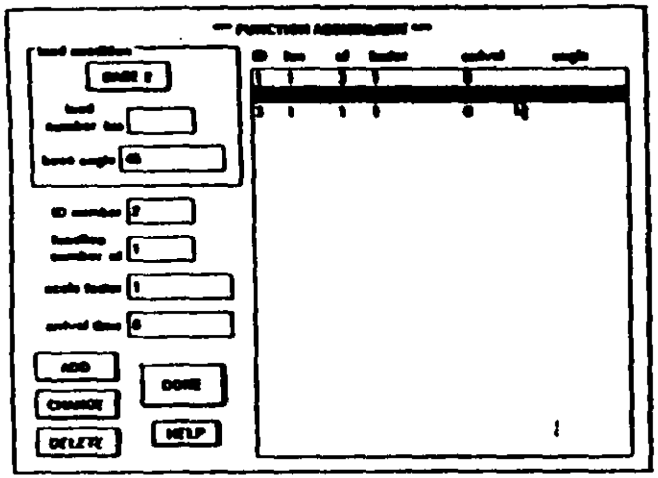
As in the CURVE Functions, you can have EQUAL or UNEQUAL files. The EQUAL files do not have time values, so you MUST enter the interval dt. The UNEQUAL files have time values for each point so that dt is disregarded.

File selection works in the same way as **Open** in the **File** menu which is described at the beginning of this chapter. Once you have a list of files that contains the desired file, click on the file in the list box so it is highlighted, be sure that the interval is set correctly, and click on SELECT. Remember that the file is not read or checked, but only the name and the interval for EQUAL files are saved.

Back at the Function Definition dialog box again, you will see that the present type is the file you have selected. Use the ADD and CHANGE buttons to make up the list of Functions in the same way as for CURVE Functions.

When the list of Functions is complete, click on DONE to return to the Time History dialog box.

In the Time History dialog box, clicking on ASSIGN will bring up the following dialog box:



You can now assign the Functions that have been defined to the Load Conditions or as base accelerations. Click on the push button in the LOAD CONDITION group to select LOAD or BASE 1,2,z. If LOAD is selected, enter the load number. IF BASE is selected, enter the base angle.

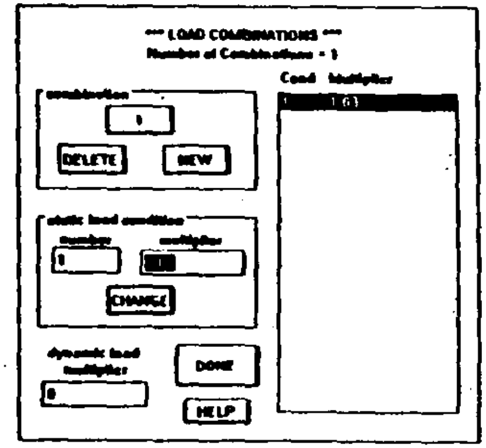
Now enter the data in the remaining edit boxes. The ID identifies the assignment and must be consecutive from one. Click on ADD to add an assignment to the list. Click on an existing assignment in the list box to bring its values into the edit boxes. You can then

change the values and click on CHANGE to change the existing assignment. Clicking on DELETE will delete the existing assignment.

Load Combination

A Load Combination consists of the set of Static Load Conditions with multipliers for each and a Dynamic Load (Response Spectrum) Multiplier. See the SAP90 Users Manual "Input Data - COMBO Block" for a complete description of Load Combinations.

Clicking on **Load Combination** brings up this dialog box:



Click on NEW to add a new Load Combination. The total number of Combinations is shown under the title at the top of the dialog box. Click on the top push button in the COMBINATION group to select the present Combination. Clicking on DELETE will delete the present Combination.

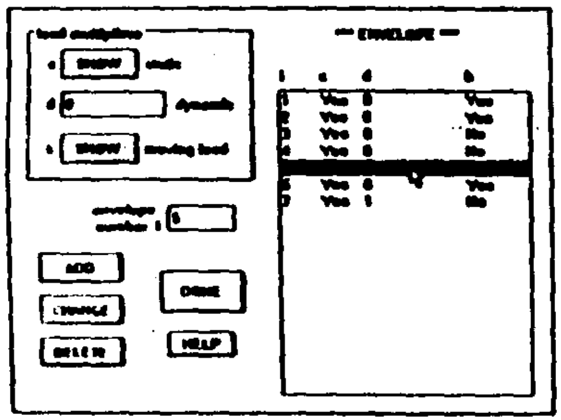
The list box will show the Static Load Conditions with multipliers for the present Combination. Click on any existing Load Condition in the list box to bring its multiplier into the edit box. Change the multiplier and click on CHANGE to change the value in the list.

Enter the Dynamic Load (Response Spectrum) Multiplier for the present Combination.

Envelope

This allows you to define Envelope combinations of moving load cases with static load conditions and the response spectrum dynamic load condition. Please refer to the SAP90 "Structural Analysis Users Manual" or the SAP90 "Bridge Analysis Users Manual", Input Data Options, ENVELOPE data block. Though primarily meant for moving loads, Envelope can be used with other loadings as well.

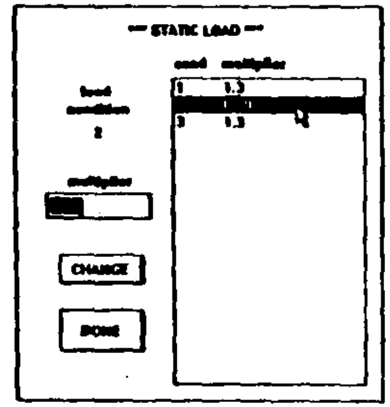
Clicking on **Envelope** brings up this dialog box:



For each Envelope, you need to specify the static multipliers (as many as static Load Conditions), the dynamic multiplier and moving load multipliers (as many as moving load Cases).

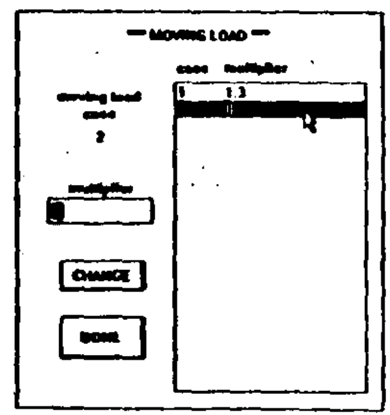
To set the static and moving load multipliers, see the next page. Then enter a dynamic multiplier and click on ADD to add a new Envelope. If you click on an existing Envelope in the list box, its multipliers will be brought into the load multipliers group.

Clicking on the SHOW static button in the Envelope dialog box shows this dialog box:



Enter a multiplier for each Load Condition and click on DONE.

Clicking on the SHOW moving load button in the Envelope dialog box shows this dialog box:



Enter a multiplier for each moving load Case and click on DONE.

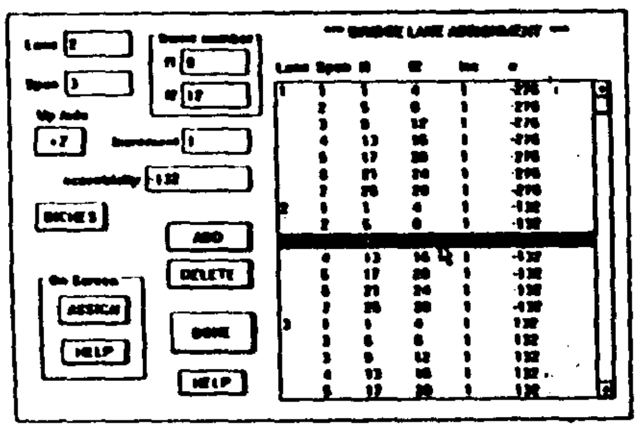
Bridge

The folowing three menu items, **Lane**, **Vehicle** and **Moving Load**, are all described in the SAP90 "Bridge Analysis Users Manual". These items allow you to analyze bridge structures for the weight of moving vehicle loads.

Lane

Lane allows you to define the bridge traffic lanes, their eccentricities with respect to the Frame element and spans. See the SAP90 "Bridge Analysis Users Manual", Input Data Options, BRIDGE Data Block, for a description of these terms.

Clicking on **Lane** brings up this dialog box:



You can add new Lanes and spans either by entering values in this dialog box or by assigning Lanes on screen. You can ONLY delete Lanes and spans by using the DELETE button in this dialog box.

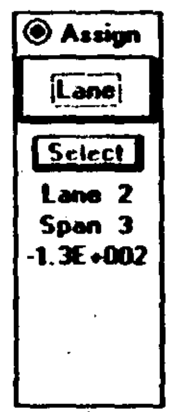
Also, you can choose the Up Axis, the global axis direction which points up in your model. All moving loads act opposite to this axis on the lane elements.

Off Screen:

Enter the frame numbers, increment and eccentricity, then set Lane and span and click on ADD to add a new Lane/span line in the list box. If a line is selected in the list box that has the same Lane/span as the edit boxes, then the ADD will occur AFTER the selected line. Otherwise, the ADD will occur at the END of the Lane/span that has the same Lane/span as the edit boxes. If there is no existing Lane/span the same as the edit boxes, then one will be created.

The list box is always sorted by Lane and then span. The Frame numbers are listed in the order entered.

Click on a Lane/span line in the list box and then click on DELETE to delete that Lane/span line.



On Screen:

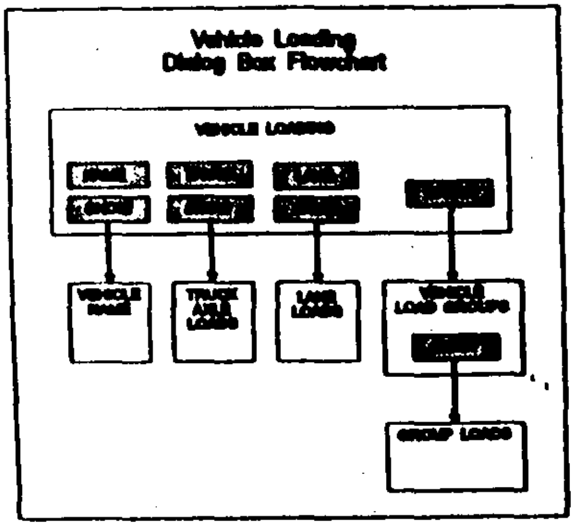
Click on the On Screen ASSIGN button and the display will appear. You can click on Select in the Control Panel (shown at left) to set the lane, span and eccentricity. On Screen assign is always in the ADD mode, because you can not delete On Screen.

To assign a Lane/span, you MUST have a set of Frames that are connected end to end. Click on the starting Joint of the starting Frame and then click on all the remaining Joints in the set of Frames in sequence. Finally, click again on the last Joint of the last Frame.

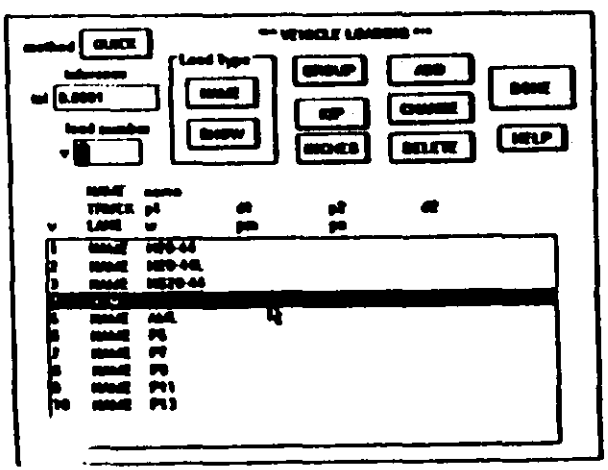
This will add one or more Lane/span lines. The Lane will be set as you specified, but there may be more than one span line if the Frame numbers (nel) are not in a sequence with an even increment between them. This has no effect except to increase the number of Lane/span lines.

Vehicle

Vehicle allows you to define the Vehicle loads and the Vehicle load groups. See the SAP90 "Bridge Analysis Users Manual", Input Data Options, VEHICLE Data Block, for a description of these terms. There are several dialog boxes associated with Vehicle loading as shown below:



Clicking on **Vehicle** brings up this dialog box:



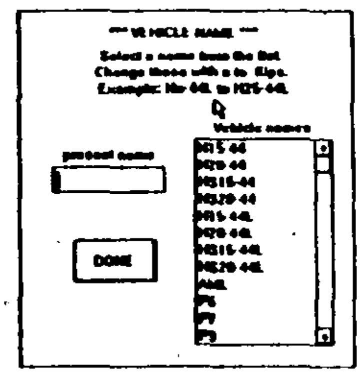
The idea here is to make a list of Vehicle loads and then assign those loads to groups. There are three types of Vehicle loads, NAME, TRUCK and LANE. These 3 types may be mixed in the list box as desired.

There is only one method and tolerance allowed for ALL loads and groups, so set them and leave them.

Select a Load Type and then click on SHOW. This will bring up one of the dialog boxes shown on the next two pages. Enter values as desired and click on DONE to return to the Vehicle Loading dialog box.

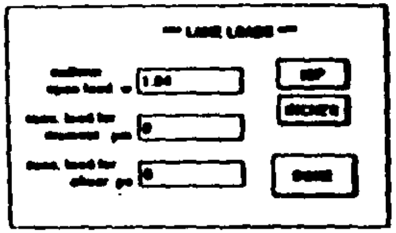
Use the ADD, CHANGE and DELETE buttons to make a list of Vehicle loads.

In the Vehicle Loading dialog box, With NAME load type, clicking on SHOW brings up this dialog box:



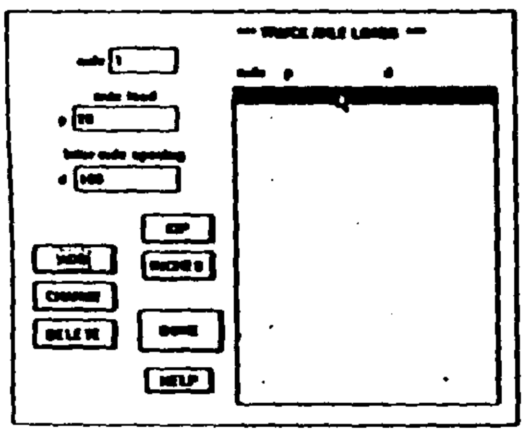
Choose a Truck name from the list. If you want a name like H30-44, click on either H20-44 or Hn-44. Then click in the edit box and use the keyboard to change the name. Click on DONE to return to the Vehicle Loading dialog box.

In the Vehicle Loading dialog box, With LANE load type, clicking on SHOW brings up this dialog box:



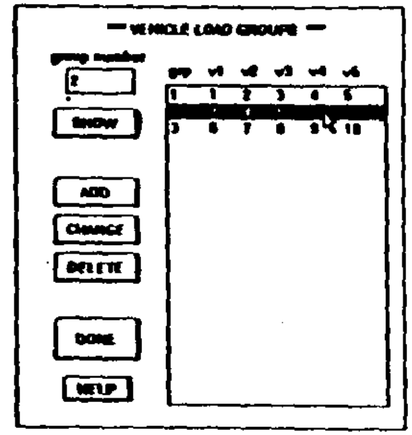
Enter values as desired and then click on DONE to return to the Vehicle Loading dialog box.

In the Vehicle Loading dialog box, with TRUCK load type, clicking on SHOW brings up this dialog box:

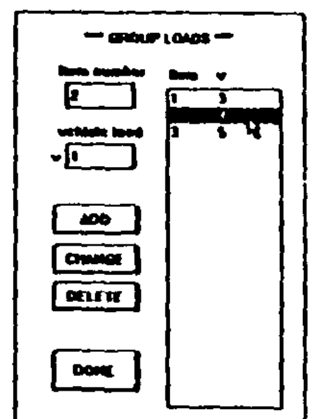


Here you can define any kind of Truck, by entering the axle load and spacing. Use the ADD, CHANGE and DELETE buttons to make up the list. Finally, click on DONE to return to the Vehicle Loading dialog box.

Once you have made a list of Vehicle loads, then you can assign them to groups. Click on the GROUP button in the Vehicle Loading dialog box to bring up this dialog box:



You can have more than five loads in a group, but only the first 5 are shown in the list box. To edit the list of loads for a group, click on that group in the list box and then click on SHOW which will bring up this dialog box:

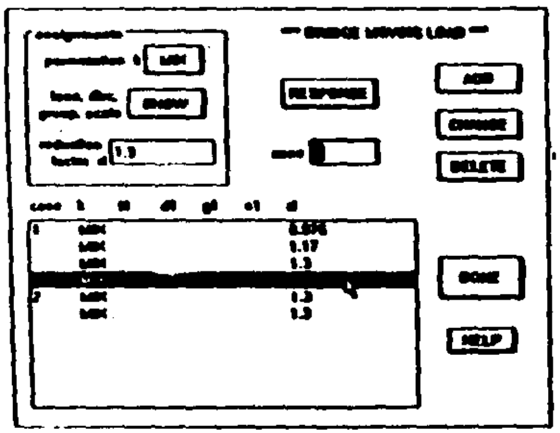


Use the ADD, CHANGE and DELETE buttons to make a list of Vehicle loads. Then click on DONE to return to the Vehicle Load Groups dialog box. Finally, use the ADD, CHANGE and DELETE buttons to make up a list of Groups.

Moving Load

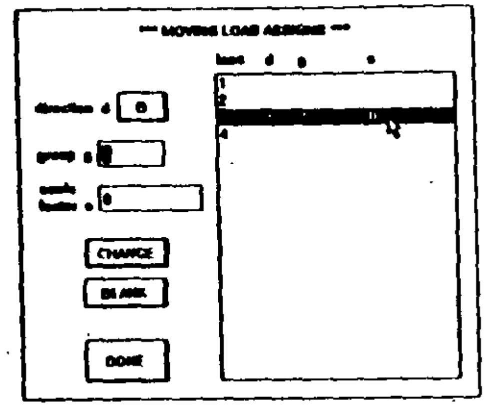
Moving Load allows you to create moving load cases that assign the vehicle load groups to the traffic lanes. See the SAP90 "Bridge Analysis Users Manual", Input Data Options, MOVING LOAD Data Block, for a description of these terms.

Clicking on **Moving Load** brings up this dialog box:



Build a list of Moving Load cases, each with one or more assignment lines. Each assignment line can have a set of directions, groups and scale factors, up to the number of Lanes. Only those values for lane 1 are shown in the list box. The assignments are automatically ordered by case in the list box, but adding an assignment line makes it the last line in that case.

Clicking on the assignments SHOW button brings up this dialog box:

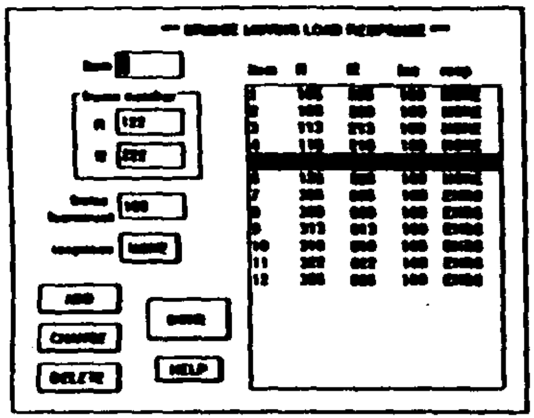


The list box shows all lanes. Lanes shown blank have no assignment.

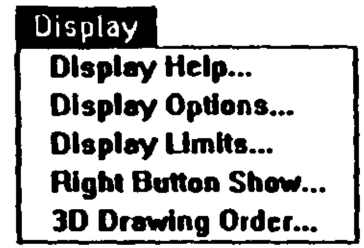
Click on a Lane in the list box, set the direction, enter group, scale factor and click on CHANGE to set new values for that Lane. Click on BLANK to remove a lane assignment. Click on DONE to return to the Moving Load dialog box.

In the Moving Load dialog box, set the permutation and enter a reduction factor. Then click on ADD to add a new assignment. Click on an assignment in the list box and then click on DELETE to delete it or enter new values and click on CHANGE to change it.

Clicking on RESPONSE in the Moving Load dialog box brings up this dialog box:



Set the frame numbers, frame increment and response and then use the ADD, CHANGE and DELETE buttons to construct a list of responses. Clicking on an item in the list box will bring its values into the edit boxes. Click on DONE to return to the Moving Load dialog box.



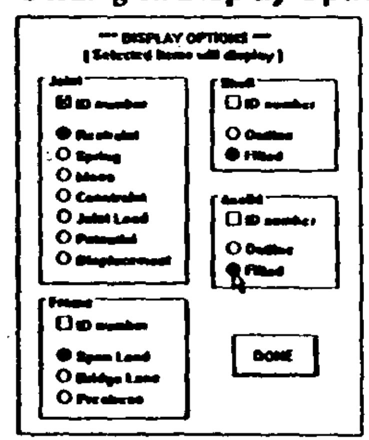
The **Display** menu items expand the display portion of the Control Panel to give additonal control over what is shown in the structure display. It also allows you to display additional information about assignments by using the Right mouse button.

Display Help

Clicking on **Display Help** brings up a dialog box with abbreviated assistance about using the other Display menu items.

Display Options

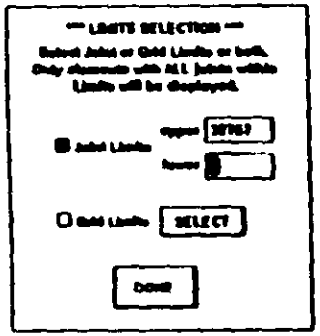
Clicking on **Display Options** shows this dialog box:



Here you can set the ID numbers of joints and assigned elements to show or not. For Joints and Frames, only ONE type of assignment can be shown at a time. Please note that if you make an assignment of a type that is not set to show, then that type will be selected and the other type deselected. This is done so that the type you are assigning will always show.

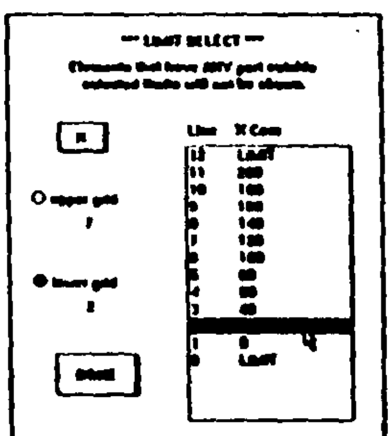
Display Limits

Clicking on **Display Limits** brings up this dialog box:



This dialog box allows you to limit the number of joints and elements shown on the display. You can limit by joint ID number or by grid planes or by both. Check Joint Limits and/or Grid Limits as desired. Note that the limits set ONLY apply when Limits is clicked in the Control Panel. Also, elements are shown only when ALL of the element joints are within the limits.

Enter the desired joint ID numbers in the edit boxes. To set the grid limits, click on SELECT, which will bring up this dialog box:

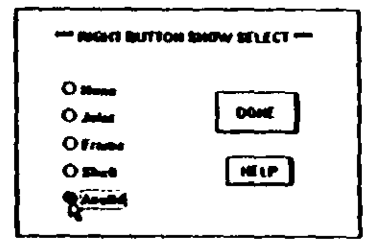


On the display, all joints and elements will be shown that are ON or ABOVE the lower grid and BELOW the upper grid. You can set each axis (X, Y or Z) independently. Click on the button to select the desired axis. Upper grid is a more positive grid value than lower grid.

Click on the upper grid radio button and then click on a line in the list box. Click on the lower grid radio button and then a line in the list box. If you want everything to show, set both to LIMIT.

Right Button Show

Clicking on Right Button Show shows this dialog box:



When the Assign button is selected in the Control Panel, then you can use the RIGHT mouse button to display information about a Joint and an element. The procedure is exactly the same as used to assign the Joint or element, except that the RIGHT button is used instead of the left. In other words, pretend to assign the same type of element right over the existing element. In the case of Asolids, you do not need to click on midside Joints a second time, but you do need to select a center Joint if there is one.

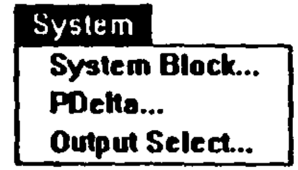
You can only select one type of element to be shown at one time. If you select None, the Joint number only will be shown in the upper right corner when you click on a Joint with the RIGHT button.

3D Drawing Order

When the 3D view is drawn on the screen, it can be drawn for maximum speed or maximum accuracy. Clicking on **3D Drawing Order** brings up a dialog box allowing you to select one or the other.

If you select maximum speed, then the Joints and elements are drawn in the most convenient order for speed, but items farthest away may appear in front and vice-versa.

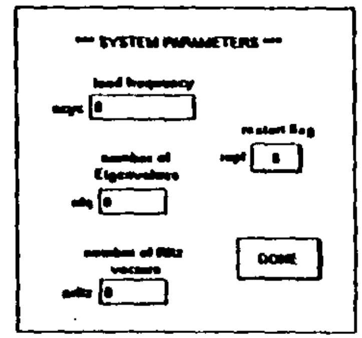
If you select 3D ordering, then elements and Joints are drawn in order from those that are farthest away and finishing with those that are nearest. This causes the nearest items to appear in front and gives the most accurate 3D view. It is also much slower if there are many Joints and elements.



The System menu contains items that apply to the entire structure and do not use the display or Control Panel.

System Block

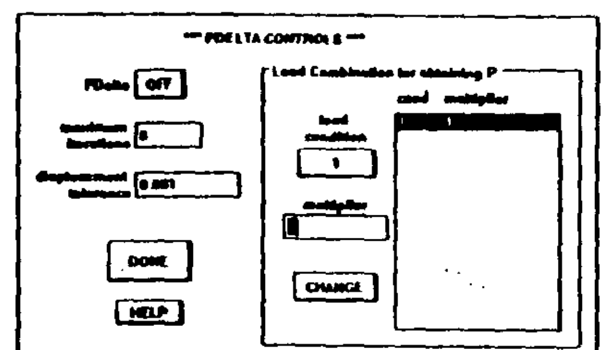
Clicking on **System Block** brings up this dialog box:



Enter values as desired. Note that nfq or nritz must be set before doing Time History or Response Spectrum loading. Click on DONE when finished. All of these items are in the SAP90 SYSTEM data block.

PDelta

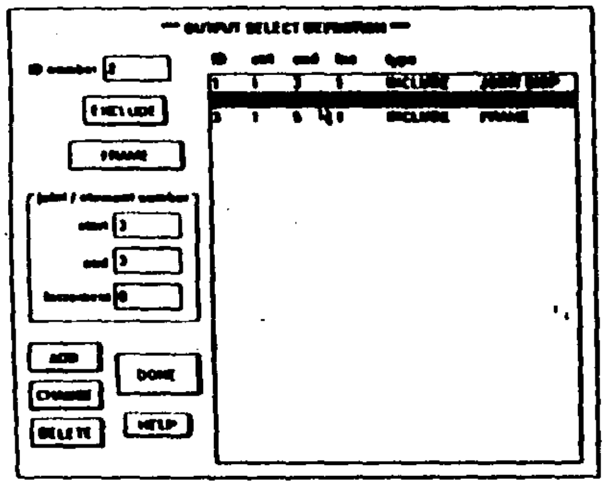
Clicking on **PDelta** shows this dialog box:



See the SAP90 Users Manual, "Input Data - PDELTA Data Block", for a description of PDelta.

If desired set PDelta ON, set multipliers for each load condition, enter maximum iterations, displacement tolerance and click on DONE.

Output Select



Clicking on **Output Select** brings up this dialog box:

This dialog box allows you to specify the joint displacements and reactions, and the element forces and stresses that will be output when you run SAP90. It has no effect in SAPIN. See the SAP90 Users Manual, "Input Data - SELECT Data Block".

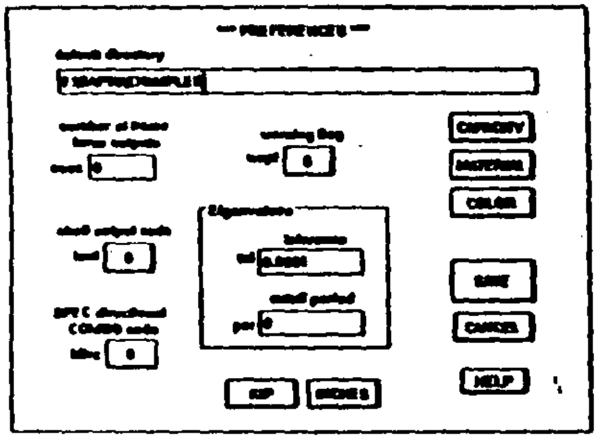
The idea is to build up a list of joints and elements that will be included or excluded from the SAP90 output. If you have ANY items in the list, then ONLY those items will be considered. If you have NO items in the list, then ALL joint displacements and reactions, and element forces and stresses will be output.

Click on the INCLUDE/EXCLUDE button, click on the output type button below it to select the type, enter the range in the edit boxes, enter the ID and click on ADD to add (insert) a item in the list. The ID must be consecutive from one.

Click on an existing item in the list to bring it into the edit boxes. Then click on DELETE to delete it, or change some values and click on CHANGE to change it in the list.

Preferences **Preferences** consist of a number of items which can be saved in a file called SAPIN.INI and then are restored when you start the program or do **New** in the **File** menu.

Click on **Preferences** to bring up this dialog box:

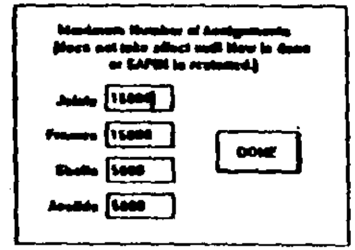


You can enter a default directory, which is the directory that will be displayed when you do an **Open** or **Save as** in the **File** menu. To set the default directory, position the cursor in the edit box and enter the directory name desired, which SHOULD include the drive. If you do not include the drive, then the directory MUST be on the same drive as SAPIN.EXE. The directory name will be checked to see if it exists when you click on the SAVE button.

For a description of nsec, refer to the SAP90 Users Manual, "Input Data - FRAME Data Block". The parameters wopt, tol and per are in "Input Data - SYSTEM Block".

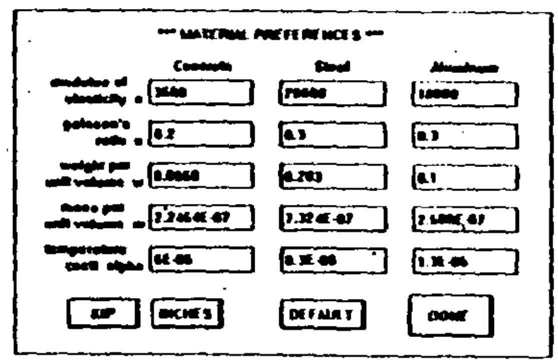
Click on the units buttons to set the units you want.

Click on the CAPACITY button to get this dialog box:



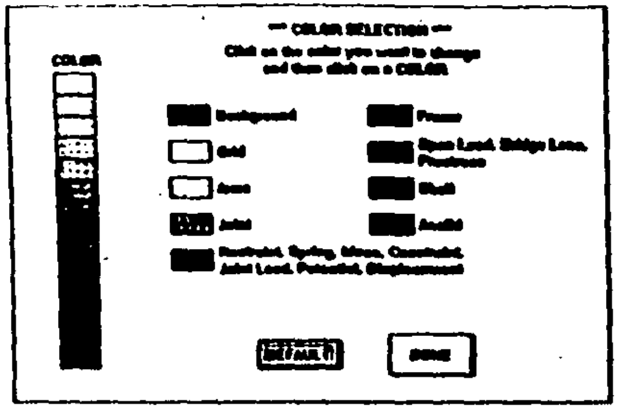
You can set values between 500 and 32767 for any item. These numbers are the maximum number of assignments so you may have any number smaller than this. Setting these numbers smaller makes the files containing the assignments smaller, which is helpful when disk space is limited.

Click on MATERIAL to get this dialog box:



These values will be available in the Frame, Shell and Asolid dialog boxes found in the **ElemTable** menu. In those dialog boxes, when you select one of the names, Concrete, Steel or Aluminum, these values will be used.

Click on the COLOR button to bring up this dialog box:



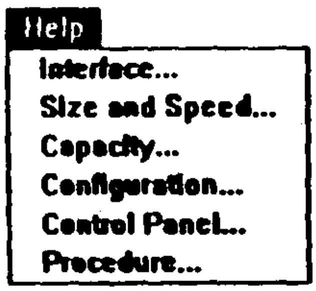
This dialog box allows you to set the colors used for the SAPIN screen when displaying grids and structural elements. Click on the button next to the items that you want to change and then a color from the COLOR set. Be SURE that you do not set an item to be the same color as the Background unless you want the item to be invisible! Note that MAGENTA is always used for presently selected structural elements and is not available otherwise.

Clicking on the DEFAULT button will change the colors to a set of colors defined in SAPIN. This is convenient if you set colors that are not usable and want to start over again. Also, the colors of the different elements described in the **Assign** menu section are all default colors.

The colors that are set will take effect when you click on DONE and remain in effect until a **New** is done.

Now click on the SAVE button in the Preferences dialog box to save the default directory, parameters, capacity, materials, units and colors in the file SAPIN.INI, which is read in each time SAPIN is started or **New** (in the **File** menu) is done.

The parameters wopt, tol, per and nsec, are saved in files when you do **Save** or **Save as**. This means that **Open** will overwrite existing parameters.



The **Help** menu includes several abbreviated help messages which apply to the entire SAPIN program. Some of this information, for example Size and Speed, is specific to the version of SAPIN that you are using and is not included in this manual. Click on the menu items to bring the help dialog boxes.

V.

HINTS and TIPS

This chapter includes a number of items to help you work faster and easier. They are ordered by importance.

Grids

Grids are useful in any structure. Even though it is possible to model a structure without them, their use is highly recommended. They allow you to model the structure in 2D in any XY, XZ or YZ plane located at a grid location. They allow partial views of the structure which can be sectioned between any two grids by using Display Limits under the Display menu and in the Control Panel. They also make choosing the 3D view angles much easier. You only need a few grid lines on each axis and they don't affect your structure in any way.

Editing

The files that SAPIN opens, edits and saves are SAP90 Input files. They can be edited with any text editor (use plain ASCII mode only). Some things are much easier to do with the editor than in SAPIN (and vice-versa). A complete description of the Input file format is in the SAP90 Users manual. Certain options of the program for instance SOLID elements and heat transfer analysis related items are currently only available through the text files. Files containing these options, however, can be read in and written out by SAPIN without disturbing the data for these options. These ˋes must be saved in the same units they have been opened

Generations

The single most important thing about generations is that they are based on proper numbering. This means that a little work with pencil and paper before you start a large structure really helps. The Joint numbering is the basis for all element generations, so it is the most important. If you have repetition, like stories of a building, number each story the same except increment by 100 or 1000 for each story. Rough the structure in before doing all the details so you can see that the generations work.

Generations and Editing

Some type of generations are easier to do by editing the Input file. If you have a building with many stories, number each story the same except increment by 100 for each story. Do one story in SAPIN and then use a text editor to copy the joints and elements for each story. Then change the numbers and the elevation. The more generations you use on the initial story, the easier this is.

Units

Different types of input appear more familiar in different set of units. For instance span loads may appear familiar when expressed in pounds/foot, material strengths when expressed in psi, etc. It is advisable to use different set of units when inputting these quantities. The program will always convert them to consistent units when saving files. It is important that the file always be opened in the same units as it was saved in. It is also important that the file always be saved in the same length units as any AISC.DAT file or other user defined section property file if one is used.

VI.

BIBLIOGRAPHY

1. **Wilson, E. L. and Habibullah, A.**

   "SAP90 - A Series of Computer Programs for the Finite Element Analysis of Structures - Structural Analysis Users Manual," Computers and Structures, Inc., Berkeley, California, 1992.

2. **Wilson, E.L. and Habibullah, A.**

   "SAP90 - A Series of Computer Programs for the Finite Element Analysis of Structures - Bridge Analysis Users Manual," Computers and Structures, Inc., Berkeley, California, 1991.

3. **Microsoft Corporation**

   "Microsoft Windows Users Guide, Version 3.0," Microsoft Corporation, 1991.

Index

D

E

F

G

H

J

K

L

M

N

O

P

Q

R

S

T

U

W

X

Y

Z

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

CURSOS ABIERTOS

XXI CURSO INTERNACIONAL DE INGENIERIA SISMICA

- 1 9 9 5 -

MOD. II ANALISIS ESTATICO Y DINAMICO DE ESTRUCTURAS

SUJETAS A SISMO

C S I SAPO™ ETABS® SAFE™ COMPUTER SOFTWARE FOR STRUCTURAL & EARTHQUAKE ENGINEERING INSTALLATION GUIDE

M. EN I. RAMON CERVANTES BELTRAN

Palacio de Minería Calle de Tacuba 5 Primer piso Deleg. Cuauhtémoc 06000 México, D.F. APDO. Postal M-2285
Teléfonos: 512-8955 512-5121 521-7335 521-1987 Fax 510-0573 521-4020 AL 26

COMPUTERS &
STRUCTURES
INC.®

SAP90™ ETABS® SAFE™

Computer Software for
Structural & Earthquake Engineering

Ramón Cervantes Beltrán

Installation Guide

Developed and written in U.S.A.

SAP90™
STRUCTURAL ANALYSIS PROGRAMS
VOLUME I

S &
ES
IC.

COMPUTERS &
STRUCTURES
INC.

COPYRIGHT

The computer programs SAP90, ETABS and SAFE and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the programs or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
(510) 845-2177

Revised June, 1991

COMPUTERS & STRUCTURES INC.

SAP90™ ETABS® SAFE™

Computer Software for
Structural & Earthquake Engineering

Installation Guide

Developed and written in U.S.A.

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
(510) 845-2177

Revised June, 1991

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP90, ETABS AND SAFE. THE PROGRAMS HAVE BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAMS, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAMS.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAMS AND MUST INDEPENDENTLY VERIFY THE RESULTS.

TABLE OF CONTENTS

Ramón Levante Beltrán

I.

INTRODUCTION

This guide provides information on customizing and installing the Computers and Structures, Inc. programs SAP90, ETABS and SAFE and their pre- and post-processors on MS-DOS/PC-DOS personal computers such as the IBM AT, PS/2 or compatibles.

The regular versions of the programs require the computer to have at least 640K bytes of RAM, a math coprocessor and a hard disk drive. The PLUS versions of the programs require a 80386 based computer with at least 2M bytes of RAM, a 80387 math-coproccesor and a hard disk drive. The graphics programs require a graphics adapter and a printer for hard copies.

Chapter II provides information for installing the programs; Chapter III provides additional requirements for use of the Windows-based interactive model building programs; and Chapter IV provides information on configuring the computer to enhance program performance.

II.

INSTALLING THE PROGRAMS

This section provides information on installing the programs SAP90, ETABS and SAFE and their pre- and postprocessors.

User familiarity with MS-DOS is assumed.

Note:

> **The characters <CR> appear repeatedly in the text of this guide. These characters mean "press the carriage return key". DO NOT type the characters <, C, R and >.**

A complete program package includes:

1. This installation guide.

2. Program manuals.

3. Program disks containing program executables, sample files and in the case of SAP90 and ETABS programs, a database of steel section properties.

4. A hardware copy protection device.

The following steps will install the program:

A. Backing up the Master Disks

Before installing the programs on the computer, make backup copies of the master disks and store the originals in a safe place. The DOS **DISKCOPY** command can be used for this purpose. Consult the DOS manual for the use of this command.

B. Copying Programs to the Hard Disk

Copy all disks to the hard disk, one disk at a time, as follows:

Place disk #1 in drive A. From the DOS C prompt and from within the subdirectory to which the program is to be copied, enter the command:

C> COPY A:*.* C: <CR>

This will copy all of the files from the floppy disk to the hard disk. After copying is complete, remove the original disk.

Repeat the same procedure for all other disks.

It is recommended that executables associated with each system (i.e. SAP90, ETABS, or SAFE) be copied to a different subdirectory and the DOS **PATH** command be used to access them.

For the SAP90 and ETABS programs, the database of AISC steel section properties is supplied in two different units. File AISC.INC contains the database in inch units and file AISC.MET contains the database in meter units. The user should copy the file with the appropriate units to a file called AISC.DAT which the programs access. This should be done from the subdirectory in which the files reside by entering:

C> COPY AISC.INC AISC.DAT <CR>

for the inch units database, or by entering:

C> COPY AISC.MET AISC.DAT <CR>

for the meter units database.

The AISC.DAT file should reside in the same directory as the SAP90/ETABS programs.

C. Using the SETUP Program

The SETUP program and associated files provided on the SETUP disk allow users to customize the company name on the program output and to configure the graphics programs for a particular graphics adapter/monitor and for a particular printer. All programs come with default settings and may be used without using the SETUP program.

To use the SETUP program, copy the contents of the SETUP disk to the hard disk as per section II-B above. The SETUP disk should be copied into the directory where the executable to be modified exists.

To begin SETUP, enter the following command from the directory to which the SETUP disk was copied.

C> SETUP <CR>

The program will respond with the following screen:

```
COMPUTERS AND STRUCTURES, INC.
SETUP Version 3.0, June, 1991

Specify type of installation to be done

   1    Insert name in program
   2    Change monitor type and graphics adapter
   3.   Change printer for hardcopy
   X.   Exit

ENTER SELECTION
```

This is the SETUP control menu. The selections 1, 2 and 3 will display lower level menus for setting other options; entering a X will exit the program.

The following three sub-sections correspond to the three options of the SETUP control menu and describe the program actions corresponding to other options.

1. Insert Name in Program

This is selection 1 on the SETUP control menu. This selection responds with the following menu:

```
Specify Program for name insertion

   1.   SAP90.EXE
   2.   ETABS.EXE
   3.   SAFE.EXE

   <cr> To Return

ENTER SELECTION
```

Selecting 1, 2 or 3 will prompt with the following message:

```
Enter name to be inserted in program (maximum of 68
characters)
```

Specifying the company name and pressing <CR> will then insert the specified name in the corresponding program and a message to that effect will be given.

Entering <CR> at the selection prompt will return user to the SETUP control menu.

The company name in the programs may be changed any number of times. However, before changing the name, the program SAP90.EXE, ETABS.EXE or SAFE.EXE, as required, must be copied from the original disk or its backup to the hard disk.

2. Change monitor type and graphics adapter

This is selection 2 on the SETUP control menu. This selection responds with the following menu:

```
Specify Graphics Program for device Installation

  1    SAPLOT.EXE
  2.   PLOTTER.EXE
  3    SAFEPL.EXE
  4.   [illegible]
  5.   SAPSTEC.EXE
  6    SAPMGA.EXE

  <cr> To Return

ENTER SELECTION
```


Selection 1, 2, 3, 4, 5 or 6 will prompt with the following choices:

```
Specify type of monitor and graphics card

  1.   Color Graphics Adapter
  2.   Enhanced Graphics Adapter (64K memory)
  3.   Enhanced Graphics Adapter (128K memory)
  4.   Video Graphics Array
  5.   HERCULES Monochrome Adapter
  6.   AT&T 6300 High Resolution Graphics

  <cr> To Return

ENTER SELECTION
```

Entering a selection will then configure the graphics program for the particular adapter and a message to that effect will be given. A <CR> without a selection will return to the SETUP control menu.

The default is the "Enhanced Graphics Adapter (128K memory)".

The graphics program adapter configuration may be changed as many times as required.

The choices shown above may vary with different versions of the program.

3. Change Printer for the Graphics Hard Copy

This is selection 3 on the SETUP control menu. This selection responds with the following menu:

```
Specify Graphics Program for device installation

   1    SAPLOT.EXE
   2.   PLOTTER.EXE
   3    SAFFPL.EXE
   4    SAPTIME.EXE
   5.   SAPSPEC.EXE
   6.   S??VIA.EXE

   <cr> To Return

ENTER SELECTION
```


Selecting 1, 2, 3, 4, 5 or 6 will prompt with the following choices:

```
Specify the type of PRINTER that will be used for graphics hardcopy

   1.   IBM graphics compatible (most printers)
   2.   Epson LQ1000 or LQ800 printers
   3.   HP Laser Jet Series II

   <cr> To Return

ENTER SELECTION
```

Entering a selection will then configure the graphics program for the particular printer and a message to that effect will be given. A <CR> without a selection will return to the SETUP control menu.

The default printer is the "IBM graphics compatible printer".

The graphics program printer configuration may be changed as many times as required.

The choices shown above may vary with different versions of the program.

D. Modifying the CONFIG.SYS file

The SAP90 and ETABS programs require that FILES=15 or more be present in the CONFIG.SYS file in the root directory of the computer before it is booted.

If the CONFIG.SYS file exists and contains FILES=15 or more it need not be modified.

If the CONFIG.SYS file exists but does not contain the Files=... line or has a number less than 15, it should be modified to contain FILES=15 by use of a text editor.

If the CONFIG.SYS file does not exist, the file CONFIG.CSI provided on the SETUP disk should be copied into the root directory of the hard disk. From the DOS C prompt, enter the command:

```
C> COPY  A:CONFIG.CSI  C:\CONFIG.SYS  <CR>
```


E. Using the Copy Protection Device

The SAP90, ETABS and SAFE software are copy protected with a hardware copy protection device that is provided with the software.

The hardware copy protection device should be attached to the parallel printer port of the computer. The port to which the device is attached should be designated as LPT1. The device goes between the computer and the printer (or any data transfer switches.)

If other programs use similar devices, all of these devices can be attached in series. Also an extension cable may be used between the computer and the device.

The copy protection device does not require the printer to be connected or, if connected, to be powered.

III.

USING THE WINDOWS-BASED MODEL BUILDERS

Programs SAPIN/ETABSIN/SAFEIN are interactive, graphical, mouse driven model builders for programs SAP90/ETABS/SAFE, respectively. These programs create input files for the respective analysis programs. The use of these model generators is not mandatory, as input files can also be created with a text editor. However, these programs provide a convenient means of generating structural models and provide options for graphically editing any existing input files.

The model building programs work under the Microsoft Windows environment. The additional requirements for the use of these programs are:

a. Microsoft Windows, version 3.0 or later, running in either the standard or the enhanced mode;

b. A mouse or other pointing device supported by Windows;

c. An EGA or VGA color display and graphics adapter supported by Windows.

To use these programs the following steps must be followed:

a. Install Windows on the computer using the Windows installation instructions.

b. Copy all files from the SAPIN/ETABSIN/SAFEIN program disk to a directory on your hard disk as per Section II-B of this guide.

c. Start Windows and the SAPIN program (or ETABSIN or SAFEIN) by entering:

WIN SAPIN <CR>

This must be done from the directory where SAPIN.EXE resides. The path must include the Windows directory using the DOS PATH command. Substitute the appropriate program name in the above command.

There are several different ways to start a program under Windows. The method described above is the fastest if Windows is not running. If Windows is already running, please refer to the Windows Users Guide for other options.

d. After the program has started and the menu appears, click on HELP for assistance. Tutorials and detailed explanations of the commands are available in the program manuals for SAPIN, ETABSIN and SAFEIN.

IV.

ENHANCING PROGRAM PERFORMANCE

The regular versions of programs SAP90/ETABS/SAFE and their pre- and post-processors are designed to work in any available memory between 480K and 640K bytes. The PLUS versions of the programs must have at least 1.0M bytes of extended memory available. The larger the available memory, the faster the programs will work. Additionally, the problem capacity will be increased as memory increases. If required, available memory may be increased by removing memory resident programs and, if necessary, by modifying the CONFIG.SYS and AUTOEXEC.BAT files and then rebooting.

The performance of some portions of the programs for large problems is heavily I/O dependent. This performance can be significantly improved by using a disk caching program. The RAM used for this disk caching, however, should be above the 640K boundary (i.e. in extended memory). Please note that some disk caching programs may be incompatible with the PLUS versions of SAP90, ETABS and SAFE.

COMPUTERS & STRUCTURES INC.

SAP90™

A Series of Computer Programs for the Finite Element Analysis of Structures

Structural Analysis Users Manual

Ramón Cervantes Beltrán

by
Edward L. Wilson
and
Ashraf Habibullah

Developed and written in U.S.A.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
Phone: (510) 845-2177
FAX: (510) 845-4096

Revised May, 1992

COMPUTERS &
STRUCTURES
INC.

SAP90™

A Series of Computer Programs for the Finite Element Analysis of Structures

Structural Analysis Users Manual

Ramón Cervantes Beltrán

by
Edward L. Wilson
and
Ashraf Habibullah

Developed and written in U.S.A.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
Phone: (510) 845-2177
FAX: (510) 845-4096

Revised May, 1992

COMPUTERS &
STRUCTURES
INC.

SAP90™

A Series of Computer Programs for the Finite Element Analysis of Structures

Structural Analysis Users Manual

by
Edward L. Wilson
and
Ashraf Habibullah

Developed and written in U.S.A.

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA
Phone: (510) 845-2177
FAX: (510) 845-4096

Revised May, 1992

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP90. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

ACKNOWLEDGMENT

Thanks are due to all of the numerous structural engineers, who over the years have given valuable feedback that has contributed toward the enhancement of this product to its current state.

The authors also wish to thank their colleagues and co-workers who have participated in the development of this program.

TABLE OF CONTENTS

Ramón Cervantes Beltrán

X. DETAILS OF THE SAP90 INPUT DATA OPTIONS

XI. REFERENCES

I.

INTRODUCTION

A. The "SAP" Series of Programs

Over the past two decades the SAP series of computer programs (see References [1,2,3]), operating on mainframe computers, have established a worldwide reputation in the areas of structural engineering and structural mechanics.

These programs represent the research work conducted at the University of California, Berkeley, by Professor Edward L. Wilson over the past 25 years.

The name "SAP" was coined in 1970 with the release of the first SAP program.

In the years that followed, further research and development in the area of finite element formulation and numerical solution techniques resulted in the release of a series of SAP programs in the form of SOLIDSAP, SAP 3 and finally SAP IV.

Since they were first introduced, the SAP series of programs have been used by hundreds of engineering firms internationally, and numerous firms have spent millions of dollars in creating modified versions of the programs to meet specific needs.

Many major commercially available structural analysis programs are based upon the element formulations and numerical methods that were originally developed for SAP.

The program has acquired the status of being the most reputable and widely used computer program in the field of structural analysis.

B. The SAP80 and SAP90 Programs

The SAP IV computer program was released almost twenty years ago and it represented the state of the art at that time.

Since the release of SAP IV, major advances have occurred in the fields of numerical analysis, structural mechanics and computer technology. These advances led to the release of SAP80, the first structural analysis program for microcomputers, over a decade ago, and more recently to the release of SAP90.

SAP90 represents new technology and was written by the author of the original SAP series of programs. The program is not a modification or an adaptation of SAP IV. The element formulations, equation solvers and eigensolvers are all new.

SAP90 represents the current state of the art; it is the technology of today. The program will remain under a constant state of development in the years to come to retain this status.

The program development is being conducted in the ANSI Fortran-77 subset environment, which guarantees portability of the software from the level of the small personal computers to the large mainframe super computers. SAP90 has been designed to run equally well on personal, mini or mainframe computers.

This version of the program is designed to be used on a MS-DOS based computer system. On computers with 640K of memory and a 30 MB hard disk, the problem-size capacity is about 4,000 joints or 8,000 equations. With a larger hard disk and with versions of the programs utilizing extended memory beyond 640K, very large problems can be solved. All numerical operations are executed in full 64-bit double precision.

The program has static analysis and dynamic analysis options. These options may be activated together in the same run. Load combinations may include results from the static and dynamic analyses.

All data is input in list-directed free format. Generation options are available for convenience. Undeformed and deformed shape plotting capabilities exist for data verification of the model geometry and for studying the structural behavior of the system.

The program is built around a blocked out-of-core active column equation solver with an automatic profile minimization algorithm. The out-of-core eigensolution procedure uses an accelerated subspace iteration algorithm.

The finite element library consists of four elements, namely, a three-dimensional FRAME element, prismatic or non-prismatic, a three-dimensional SHELL element, a two-dimensional ASOLID element and a three-dimensional SOLID element. The two-dimensional frame, truss, membrane, plate bending, axisymmetric and plane strain elements are all available as subsets of these elements. All necessary geometric and loading options associated with the elements have been incorporated. A boundary element in the form of spring supports is also included.

There is no restriction on mixing or combining element types within a particular model.

Loading options allow for gravity, thermal and prestress conditions in addition to the usual nodal loading with specified forces or displacements. Dynamic loading can be in the form of a base acceleration response spectrum, or time varying loads and base accelerations.

C. The "SAP" Warning

The effective application of a computer program for the analysis of practical situations involves a considerable amount of experience. The most difficult phase of the analysis is assembling an appropriate model which captures the major characteristics of the behavior of the structure. No computer program can replace the engineering judgment of an experienced engineer. It is well said that an incapable engineer cannot do with a ton of computer output what a good engineer can do on the back of an envelope. Correct output interpretation is just as important as the preparation of a good structural model. Verification of unexpected results needs a good understanding of the basic assumptions and mechanics of the program. Equilibrium checks are necessary not only to check the computer output but to understand basic structural behavior.

Back in 1970 the original SAP publication carried the following statement:

> "The slang name SAP was selected to remind the user that this program, like all computer programs, lacks intelligence. It is the responsibility of the engineer to idealize the structure correctly and assume responsibility for the results."

The name SAP has been retained for this program for exactly the same reason.

II.

SYSTEM PREPARATION AND EXECUTION PROCEDURES

This chapter deals with the installation and execution of the SAP90 Structural Analysis program on an MS-DOS based computer system. User familiarity with MS-DOS is assumed.

The SAP90 Structural Analysis package includes:

- This SAP90 Structural Analysis Users Manual
- The SAPIN Users Manual [4]
- The SAPLOT Users Manual [5]
- The SAPTIME Users Manual [6]
- The SAPSPEC Users Manual [7]
- The SAP90 Structural Analysis Verification Manual [8]
- The SAP90/ETABS/SAFE Installation Guide [9]
- Program diskettes, containing some or all of the following:
  - SAP90 Program Executables (.EXE files); contents depends upon options acquired

- SAPIN and SAPLOT Executables
- SAPTIME and SAPSPEC Executables, if Dynamics option acquired
- SETUP Program and Associated Files
- SAP90 Sample Example Files

The number of diskettes provided with the program depends upon the modules that are acquired.

Note: the characters <CR> appear repeatedly in the text of this chapter. These characters mean "press the Carriage Return (Enter) key." Do not type the actual characters <, C, R and >.

A. Installing, Configuring and Testing

The programs provided must first be copied to the hard disk and the programs and the computer must be configured before the programs can be used. Follow the instructions in the Installation Guide [9] for this procedure.

Before putting the system into a production mode, the user should test the system by running SAP90 using the sample example input data files provided with the program. The output files produced should be compared to the corresponding output files that are also provided.

B. Preparing the SAP90 Input Data File

Before executing SAP90 the user needs to prepare the data for the specific structure that is to be analyzed. The user must first thoroughly read this manual and understand the basic assumptions of the program. The user must then prepare data in an input data file. The input data file may have any valid MS-DOS filename without an extension, and may be located in any convenient directory or subdirectory. Throughout this manual the input data filename "EXAMPLE" will be used by way of example. The user may choose a different filename for each problem to be analyzed.

The input data file may be prepared by one of two methods:

- Using the interactive data preprocessor SAPIN [4], which must be executed from within Microsoft Windows.
- Using the text editors EDLIN or EDIT (or any other MS DOS or Windows compatible editor).

The contents of the input data file is described in Chapter VII and must conform to the specifications detailed in Chapter X of this manual. Sample input data files are provided on the program diskettes (the filenames with no extensions, FRAME, WALL, etc.)

C. Executing the SAP90 Program

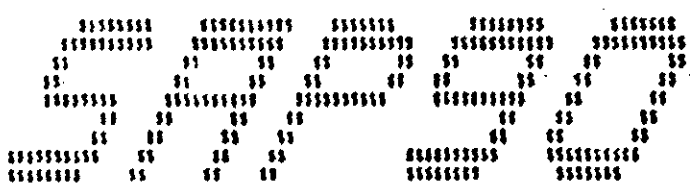
Suppose that the data associated with the problem the user wishes to analyze has been entered into a SAP90 input data file called EXAMPLE.

From the directory where this input data file is resident, enter the command:

C> SAP90 <CR>

Note: the SAP90 input data file and the SAP90 program executables must exist in the same directory unless a path to all of the SAP90 executables has been activated using the MS-DOS PATH command.

After entering the **SAP90** command the following banner will appear on the screen:



```
STRUCTURAL ANALYSIS PROGRAMS

VERSION 5.40

Copyright (C) 1970-1992
EDWARD L. WILSON
All rights reserved

                                        <CR>
```


Then enter **<CR>**

and the following will appear on the screen:

```
THIS COPY OF SAP90 IS FOR THE EXCLUSIVE USE OF

THE LICENSEE

USE OF THIS COPY BY OTHER INDIVIDUALS OR COMPANIES
IS UNETHICAL AND IN VIOLATION OF NON-DISCLOSURE
AND COPYRIGHT AGREEMENTS.

ADDITIONAL INFORMATION MAY BE OBTAINED FROM

COMPUTERS AND STRUCTURES, INC.
1995 UNIVERSITY AVENUE
BERKELEY, CALIFORNIA   94704

TEL:  (510) 845-2177
FAX:  (510) 845-4096

IT IS THE RESPONSIBILITY OF THE USER TO VERIFY
ALL RESULTS PRODUCED BY THIS PROGRAM.

ENTER NAME OF INPUT DATA FILE (EXTENSION NOT ALLOWED)
```

In response to the prompt for the input data filename enter:

EXAMPLE <CR>

The program will then enter the input phase. A series of screen messages will identify the progress of the various steps. This phase reads the input data file, checks all the data for compatibility and creates an output file, EXAMPLE.SAP. This file contains a formatted and tabulated echo of the input data. When the screen indicates completion of the input phase, the user should print this output file and thoroughly check the data for numerical correctness.

If the data is error-free, enter:

C> GO <CR>

This single command will execute the series of SAP90 program modules in the required sequence for the analysis of the structure defined by the input data file EXAMPLE.

A series of messages will continuously appear on the screen, identifying the progress of the solution. At the end of this execution sequence, the user will find a series of output files that are created by the program as described in Chapter VIII.

To print an output file the MS-DOS PRINT command may be used.

Appropriate line counts and page ejects are built into the files.

D. Saving the Screen Image

If a SAP90 job is expected to run for more than an hour, it is recommended that the printer be switched on and put on-line to obtain a hard copy of the program messages that appear on the screen during execution.

This information is helpful in tracing possible data-check and execution errors, warnings and other messages, especially if system malfunction is suspected.

The following sequence of operations will activate the creation of the hard copy:

(1) Before giving the SAP90 command, turn the printer on and it on line.

(2) Holding down the Ctrl key, press the PrtSc key.

All information that appears on the screen will now be printed.

In order to deactivate the hard-copy creation, repeat Step 2.

E. Clearing the Disk

After the execution of the **SAP90** or **GO** commands, the user will find a series of files of the form EXAMPLE.* that are not described in either Chapter VII or VIII. These are the SAP90 intermediate execution files. These files are needed if a restart run is anticipated (See Chapter IX). Also, some of these files are needed for plotting undeformed shapes, deformed shapes, mode shapes, element force diagrams or stress contours by the postprocessor SAPLOT [5], for plotting input or response time functions for time history analysis by the postprocessor SAPTIME [6], or for running the design-check postprocessors SAPSTL [10] and SAPCON [11].

If these files are not needed, a series of batch commands is available to facilitate the selective clearing of these files from the disk.

From the directory where the files are resident, entering the command

C> JUNK EXAMPLE <CR>

will erase all the SAP90 files except the input file, the output files (listed in Figure VIII-1) and the intermediate files required by SAPLOT, SAPTIME, SAPSTL and SAPCON.

Entering the command

C> EATOUT EXAMPLE <CR>

will erase only the SAP90 output files listed in Figure VIII-1.

Entering the command

C> EAT EXAMPLE <CR>

will erase all the intermediate **SAP90** files, leaving behind the data file EXAMPLE and the output files listed in Figure VIII-1. **JUNK** is a subset of **EAT**.

F. Using the Interactive Graphics Programs

The interactive graphics package SAPLOT may be used for plotting undeformed shapes, loading, deformed shapes (and mode shapes), and element force diagrams or stress contours of the structure.

Undeformed geometry and loading plots may be obtained after an error free execution of the **SAP90** command, i.e., before or after executing the **GO** command.

Static deformed shapes and mode shapes, FRAME element force diagrams, and SHELL, ASOLID and SOLID element stress contours may only be plotted after a successful execution of the **GO** command.

See Reference [..] for details associated with the options and usage of the SAPLOT postprocessor.

The interactive graphics package SAPTIME may be used for plotting input and response time functions.

SAPTIME may only be used after the successful execution of the **GO** command.

See Reference [6] for details associated with the options and usage of the SAPTIME postprocessor.

III.

SAP90 TERMINOLOGY

Data preparation for a structural analysis problem basically involves the following steps:

- Describing the geometry of the structure,
- Describing the material and section properties of the members,
- Defining the static and/or dynamic load conditions for which the structure needs to be analyzed.

This chapter deals with specific conventions and terminology used by SAP90 in the input preparation and the output interpretation phases of the analysis.

A. Joints and Elements

The basic geometric dimensions of the structu
by placing joints (or nodes) on the structure. Each joi
a unique identification number and is located in space with coordinates that are associated with a global three-dimensional coordinate system.

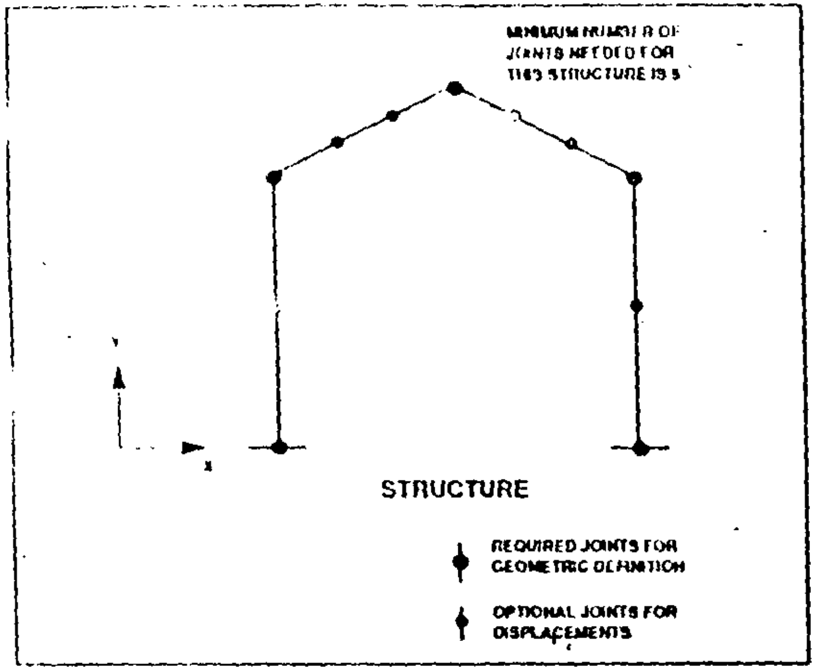


Figure III-1
Location of Joints

The structural geometry is completed by connecting the predefined joints with structural elements that are of a specific type, namely: beams, trusses, shells, plates, etc. Each element is assigned a unique identification number.

The following are some of the factors that need to be considered in locating joints on a structure:

- The number of joints should be sufficient to describe the geometry of the structure; see Figure III-1.

- Joints and element boundaries need to be located at points, lines and surfaces of discontinuity; e.g. changes in material properties, section properties, etc.

- Joints should be located at points on the structure where displacements are to be evaluated.

- Joints should be located at points on the structure where concentrated loads are to be applied, or masses are to be lumped in a dynamic analysis. FRAME elements, however, can have concentrated lateral loads applied within their spans.

- Joints should be located at all support points. Support conditions are simulated in the structure by restricting the movement of the specific joints in specific directions.

- Finite elem... meshes should be refined enough (using small elements and closely-spaced joints) to capture stress intensities and displacement variations in regions of interest. This may require changing the mesh after a preliminary analysis.

The loading may be applied to the structure as concentrated loads acting on specific joints or as distributed loads (including thermal loads) acting on the elements.

B. Global and Local Coordinate Systems

For the definition of all the input and output associated with the joints, a three-dimensional, right-handed "X Y Z" Cartesian system is used. This system is known as the global coordinate system.

The following input data is prepared with respect to the global coordinate system:

- Joint coordinates
- Joint restraints (support conditions)
- Joint spring supports
- Joint loading
- Joint constraints
- Joint specified displacements

The following output is referred to the global coordinate system:

- Joint displacements
- Joint reactions

For input associated with the elements, a three-dimensional, right-handed "1-2-3" Cartesian system is used. This coordinate system is different for each element and is known as the element local coordinate system.

The following input data is prepared with respect to the local coordinate system:

- Element material and section properties
- Element loading (unless otherwise noted)

The following output is referred to in the local coordinate system:

- Element forces and stresses (unless otherwise noted)

There are various ways in which arbitrary three-dimensional local coordinate systems may be defined; however, most programs explicitly or implicitly use vector algebra. A fundamental knowledge of the operation of vector **cross products** is very helpful in clearly understanding how local coordinate systems are generated.

It should be obvious that two intersecting lines (called vectors) will *uniquely* define a plane. To perform a cross product, two intersecting vectors are required. A cross product of two intersecting vectors yields a third vector which is in a direction normal to the plane defined by the two intersecting vectors.

For example, as shown in Figure III-2, let vector V_1 and another vector V_n define the 1-3 plane, where V_n is any conveniently defined vector in the 1-3 plane. Now, as V_2 is normal to the 1-3 plane, V_2 can be defined by the cross product of V_n and V_1.

Therefore, for vectors defined in a right-handed coordinate system,

$$V_2 = V_n \times V_1 \qquad \text{(note that } V_1 \times V_n \text{ gives } -V_2\text{).}$$

Once V_2 is defined, V_3, which is normal to the plane defined by V_1 and V_2, can be defined by the cross product of V_1 and V_2:

$$V_3 = V_1 \times V_2 \qquad \text{(note that } V_2 \times V_1 \text{ gives } -V_3\text{).}$$

In the definition of element local coordinate systems, one of the three V_1, V_2 or V_3 vectors is located using the element geom

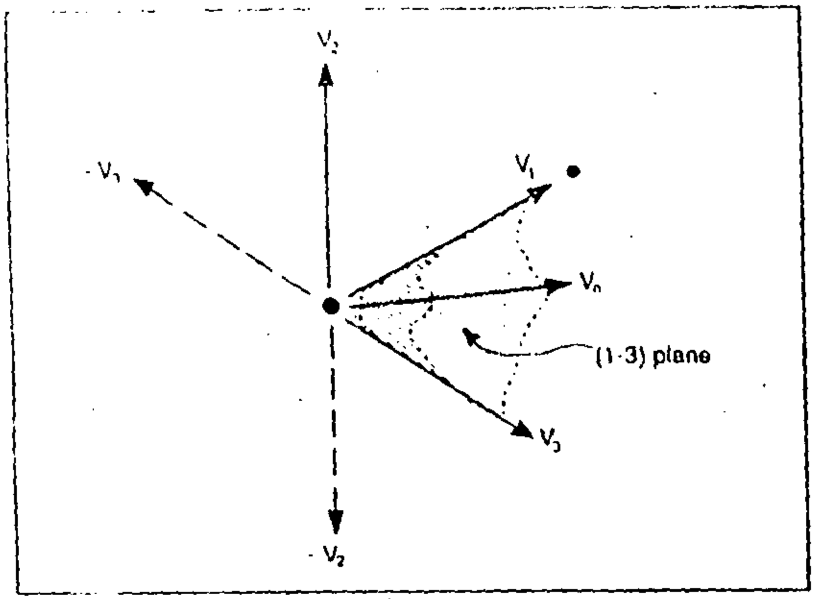


Figure III-2
Vector Cross Products

etry, the V_n vector is defined by the user, and then the V_1,V_2,V_3 set is completed using cross products as defined above.

C. Degrees of Freedom

Every joint of a three-dimensional structural model has six displacement components: the joint can translate in the global X, Y and Z directions and it can rotate about the global X, Y and Z axes. The directions associated with these six displacement components are known as the **degrees of freedom** of the joint. They will be called **UX**, **UY**, **UZ**, **RX**, **RY** and **RZ**, respectively, throughout this manual.

If the displacement of a joint along any one of its six degrees of freedom is known to be zero, such as at a support point, then

that degree of freedom is known as an **inactive** degree of freedom. Degrees of freedom at which the displacements are not known are termed **active** degrees of freedom.

In general, the displacement of an inactive degree of freedom is usually known, and the purpose of the analysis is to find the reaction in that direction. For an active degree of freedom the applied load is usually known (could be zero), and the purpose of the analysis is to find the corresponding displacement.

Every active degree of freedom in the structure has an associated equation number: if there are *N* active degrees of freedom in the structure, there are *N* equations in the system, and the structure stiffness matrix is said to be of order *N*. The number of equations may be reduced by the presence of constraints.

If there are degrees of freedom in the system at which the stiffness is known to be zero, such as out-of-plane degrees of freedom in a two-dimensional planar analysis, these degrees of freedom should be inactivated because they unnecessarily increase the size of the system of equations by introducing null components into the analysis. The user must have a basic understanding of the direct stiffness method in order to identify the degrees of freedom of a particular joint that do not have any associated stiffnesses. The stiffness components of a joint are determined by the types of elements connecting to the joint.

For example, a three-dimensional beam element contributes stiffness to all six degrees of freedom of the joint to which it connects, whereas a three-dimensional truss or solid element contributes stiffness only to the translational degrees of freedom and none to the rotational components.

The user should be able to identify the null degrees of freedom upon close examination of the types of elements framing into

Element Type	UX	UY	UZ	RX	RY	RZ
2-D FRAME (X-Y plane)	0	0	1	1	1	0
2-D FRAME (Y-Z plane)	1	0	0	0	1	1
2-D FRAME (Z-X plane)	0	1	0	1	0	1
3-D FRAME	0	0	0	0	0	0
3-D TRUSS	0	0	0	1	1	1
2-D GRID (X-Y plane)	1	1	0	0	0	1
3-D SHELL	0	0	0	0	0	0
ASOLID (X-Y plane)	0	0	1	1	1	1
ASOLID (Y-Z plane)	1	0	0	1	1	1
ASOLID (Z-X plane)	0	1	0	1	1	1
SOLID	0	0	0	1	1	1
where: 1 = null or no stiffness exists and 0 = stiffness exists						

Table III-1
Stiffness Terms for Different Types of Elements

a particular joint. Table III-1 defines the stiffness components associated with some types of elements.

If there is any doubt as to whether a particular degree of freedom will receive any stiffness from the elements, the degree of freedom should be left active.

D. Load Conditions and Load Combinations

In the interest of clarity it is important to recognize the difference between a load condition and a load combination as defined in the SAP90 terminology.

The **load conditions** are the independent loadings for which the structure is explicitly analyzed.

The **load combinations** are loadings formed by linear combination of the independent loading conditions.

Typically, all analytical results are output for each of the independent load conditions. However, if load combination data is specified, the results are for the load combinations and not for the load conditions.

E. Units

There are no built-in units in the SAP90 computer program. The user must prepare the input in a consistent set of units. The output produced by the program will then conform to the same set of units.

The fundamental units needed for a structural analysis (and the corresponding abbreviations used in this manual) are: length (L), time (T), mass (M) and temperature (H). Arbitrary units may be chosen by the user for each of these quantities. The derived quantity of force (F) must have consistent units given by $F = ML/T^2$. Note that the ratio of the weight to mass for a physical object is given by the acceleration due to gravity expressed in consistent units. Angular measurements may be in degrees (deg) or radians (rad), depending on the quantity.

For example, if the user chooses to use kips (1000 pounds) and inches as the input force and length units, all the dimensions of the structure must be entered in inches and all the loads in kips. The material properties should also conform to these units. The output units will then be in kips and inches, so that the frame member axial forces will be in kips, and all bending moments will be in kip inch units. All displacements will be in inches. Joint rotations are in radians, irrespective of units.

However, if the user anticipates using any of the design postprocessors of the SAP90 system, such as SAPSTL [10] or SAPCON [11], the SAP90 input data will need to be prepared in the specific set of units as defined in the corresponding postprocessor manual.

F. Solution Accuracy and Verification

The SAP90 equation solver monitors the accuracy of the analysis as it progresses, and produces information that warns the user if the quality of the solution starts to degrade. The procedure is very helpful in identifying numerically ill-conditioned and unstable structural systems. All arithmetic operations are performed in 64-bit double precision, which translates to an accuracy of about 15 decimal digits.

The solver calculates the loss of accuracy for each of the equations as they are reduced by determining the number of digits lost in the reduction process. On the average, this number lies between 1 and 3, but it may be as large as 7 for isolated

cases. If the loss of accuracy exceeds 11 digits the program terminates execution.

In addition to the check on the loss of accuracy during the reduction, the program performs a check on the accuracy of the complete solution. The global load vectors are recalculated by assembling the element load vectors obtained from the product of the element stiffness matrices and the element displacements. The calculated global load vectors should balance the applied nodal load vector, within a particular load condition.

G. Program Capacity

The capacity of a structural analysis program is generally defined in terms of the number of equations, number of joints, number of elements and the number of load conditions that the program can accommodate for a particular model in a run.

The personal computer version of SAP90, running under the 640K-byte limit and with a 30M-byte hard disk has been configured and tested to satisfy the following specifications.

- **Static Analysis or Steady State Analysis:**

Maximum number of equations, **neq**	8,000
Maximum number of joints, **numj**	4,000

- **Dynamic Analysis:**

Maximum number of equations, **neq**	5,000
Maximum number of joints, **numj**	2,500

There are other factors which have a bearing on the performance and capacity of the program, such as: number of elements, number of constraints, etc. However, these parameters, in general, are not controlling factors.

High coupling within the equations of the stiffness matrix of large systems will make hard-disk storage requirements critical.

The user should review the sample data cases that are provided with the complete SAP90 package for a better understanding of the capacity of the program.

Larger hard disks and available versions of the program that utilize memory in excess of 640K-byte would provide much larger capacities.

IV.

SAP90 MODELING OPTIONS

This chapter highlights the various options of SAP90 that pertain to the generation of the computer model. Details associated with these options are presented in Chapter X.

A. Joint Coordinate Generation

Extensive generation options for creating the joint coordinate data are built into the program. The following options are available:

- Linear Generation

  In this generation option, the joint data for two joints is defined. The program generates the data for equally spaced joints on the line defined by connecting the two joints. Optionally, gradually increasing or decreasing spaces (arithmetic progression) between joints along the line can also be generated.

- Quadrilateral Generation

  In this generation option, the joint data for four joints is defined. The program generates the data for equally spaced joints in two directions within the region defined by the four joints.

- Frontal Generation

  This generation option is for defining joints for a rectangular (or parallelogram) grid system. Joints are generated in two directions. The joints need not be equally spaced. This option is particularly useful in the generation of joints for a multistory building type model.

- Lagrangian Generation

  This generation option allows for the automatic generation of joints on a complex four-sided surface in space. The coordinates of all the joints defining the four sides are specified. The generated joints infill this enclosed surface, defining a well-graded system of joints, where the coordinates of each generated joint are the average of the coordinates of the joints around the generated joint.

- Cylindrical and Spherical Generation

  This generation option allows the user to generate joints on a circle. The plane of the circle may have any arbitrary orientation in space. Repetitive use of the cylindrical generation option can be conveniently used to define joints in a spherical form. Helical and spiral generation options are also available.

B. Joint Support Conditions

Free, fixed, pinned or spring support conditions are allowed at any joint. Options for providing translational and rotational spring constants are available.

The program will generate reaction forces along fixed or spring supported degrees of freedom.

C. Joint Constraints

One-dimensional global constraint options are available which enable the user to selectively equate the displacements of global degrees of freedom, resulting in a reduction of the number of equilibrium equations in the total system.

An example of the use of the constraint option is in the analysis of the two-dimensional frame shown in Figure IV-1(a). If the axial deformations in the beams are negligible, the X-displacements of all the joints at a particular level will be equal and, instead of five equations, a single equation can be used to define the X-displacement of the whole floor. However, it should be noted that this will result in the axial forces of the beams being output as zero, as the constraint will cause the ends of the beams to translate together in the X-direction. Interpretations of such results associated with the use of constraint transformations should be clearly understood.

A more practical use of the constraint option is in the connecting or merging of different segments of a model as shown in Figure IV-1(b).

Say that a model is developed in two separate segments, I and II. Joints 121 through 125 are associated with Segment I, and Joints 221 through 225 are associated with Segment II. Joints 121 through 125 share the same location in space as Joints 221 through 225, respectively. These are the interfacing joints between the two segments of the model.

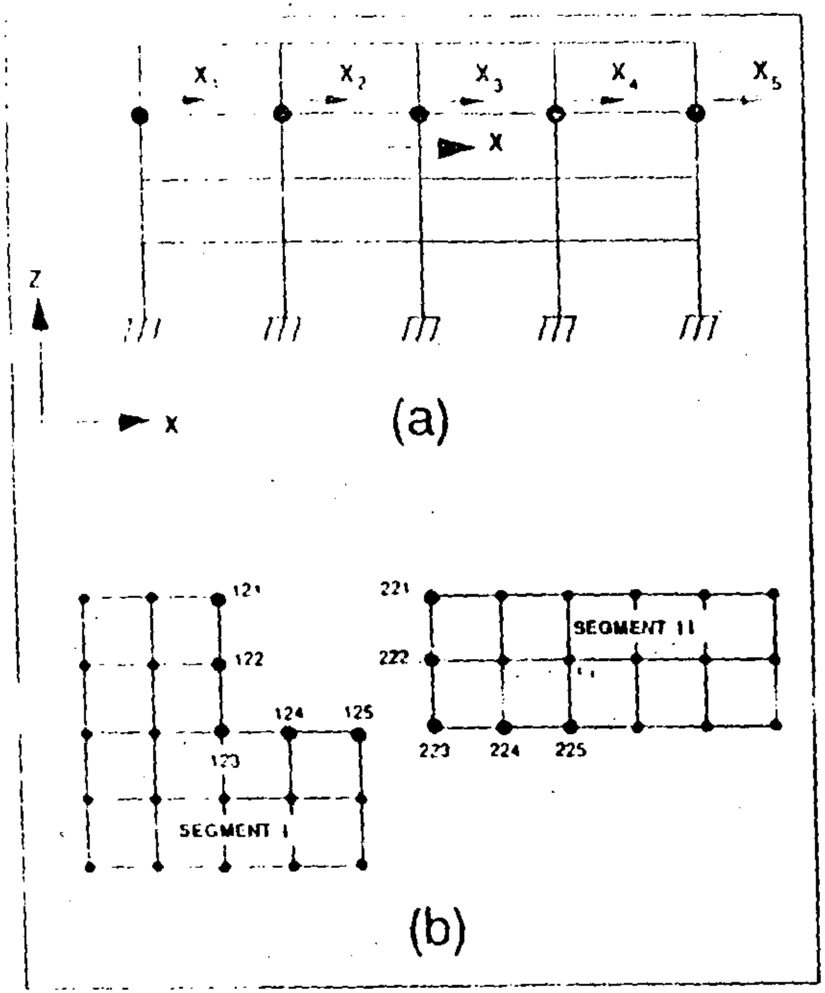


Figure IV-1
The Constraint Option

The constraint option can be used to connect the six degrees of freedom of each of the Joints 121 through 125 to the corresponding degrees of freedom of Joints 221 through 225 respectively, giving an integrated model.

D. Types of Elements

The SAP90 element library consists of basically four types of elements:

D.1. The FRAME Element

This element is for modeling:

(a) Two- and three-dimensional frame systems
(b) Two- and three-dimensional truss systems

The basic element is a three-dimensional prismatic (or optionally non-prismatic) beam-column formulation which includes the effects of biaxial bending, torsion, axial deformations and biaxial shear deformations. See Reference [17]. The non-prismatic beam-column formulation allows for linear, parabolic or cubic variation of the major moment of inertia along the length of the element and for linear variations in axial and shear areas, minor moment of inertia and the torsional constant.

The beam-column element degenerates to a truss element if the biaxial moments of inertia, the corresponding shear areas and the torsional inertia are specified as zero.

Span loading in the form of uniform loads (local and global), point loads, trapezoidal loads, gravity loads and thermal loads is allowed. Also included are options for rigid joint offsets and member end releases.

Element generation options are available.

Element forces in the element local coordinate system are produced at the ends of each element (at the face of the supports as shown on Figure X-16) and at other controlling points along the span of the element. Optionally, the user can specify the number of equally-spaced output stations along the clear length of the element.

D.2. The SHELL Element

This element is for modeling:

(a) Three-dimensional shell structures
(b) Two- and three-dimensional membrane systems
(c) Two- and three-dimensional plate bending systems

The 4-node element formulation is a combination of membrane and plate bending behavior. The membrane is an isoparametric formulation including translational in-plane stiffness components and a rotational stiffness component in the direction normal to the plane of the element. See Reference [21]

The plate bending behavior includes two-way out-of-plane plate rotational stiffness components and a translational stiffness component in the direction normal to the plane of the element. The plate bending behavior does not include any effects of shear deformation. See Reference [18].

Element loading in the form of normal pressure loading, thermal and gravity loads is allowed.

Element generation options are available.

An eight-point numerical integration formulation is used for the elements. Element stresses or resultant forces and moments, in the element local coordinate system, are evaluated at the integration points and extrapolated to the joints of the element. An approximate error in the element stresses or resultants can be estimated from the difference in values calculated from different elements attached to a common joint. This will give an

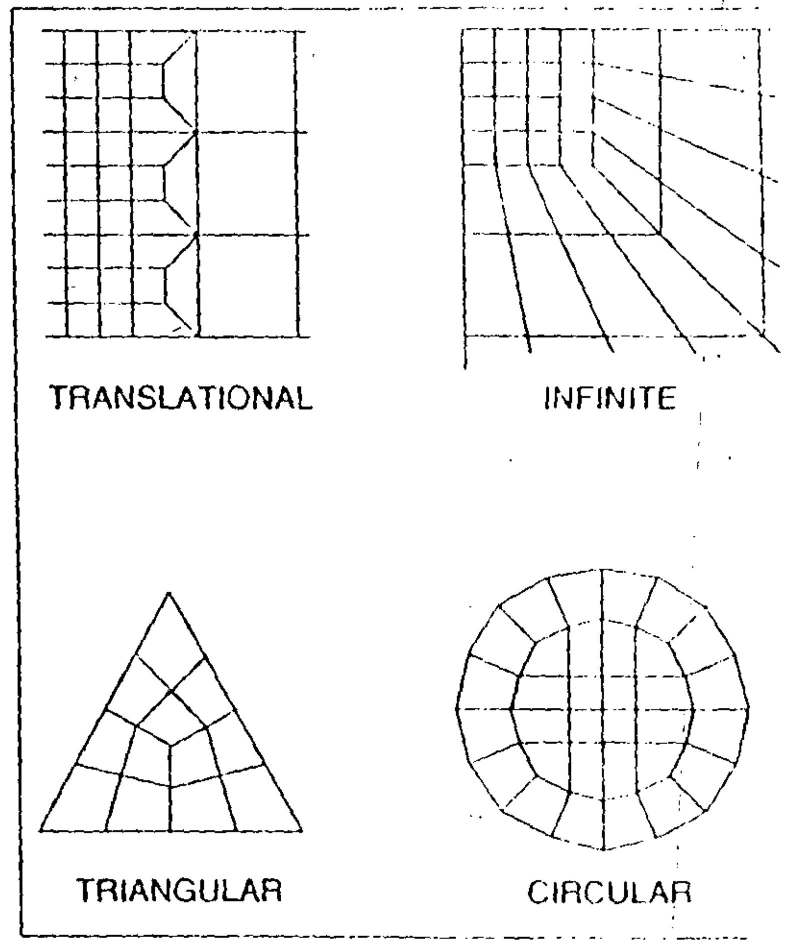


Figure IV-2
Finite Element Mesh Examples and Mesh Transitions Using 4-Node Elements

indication of the accuracy of the finite element approximation and can then be used as the basis for the selection of a new and more accurate finite element mesh.

A triangular element option is available, but is recommended for transitions only. The stiffness formulation of the 3-node element is reasonable; however, stress recovery is poor.

Typical mesh configurations and transitions using a 4-node element are shown in Figure IV-2.

D.3. The ASOLID Element

This element is for modeling:

(a) Three-dimensional plane-strain structures
(b) Two-dimensional plane-stress structures
(c) Three-dimensional axisymmetric structures (with axisymmetric loading)

This is a two-dimensional, variable 3- to 9-node element, based upon an isoparametric formulation. See Reference [20]. The element must be planar and must always exist parallel to the global principal planes (i.e., parallel to the X-Y, Y-Z, or Z-X planes).

Temperature-dependent, orthotropic material properties are allowed.

Element loading in the form of thermal, pressure gradient and gravity loads is allowed. Radial loads due to constant rotational angular velocities are possible for axisymmetric solids.

Element generation options are available.

The variable node option of the element is very useful for generating transition interfaces between coarse and fine finite element meshes. See Figure IV-3.

An eight-point numerical integration scheme is used for the elements. Element stresses in the global coordinate system are evaluated at the integration points and extrapolated to the joints [illegible] element. An approximate error in the stresses can be [illegible]ated from the difference in element stresses calculated

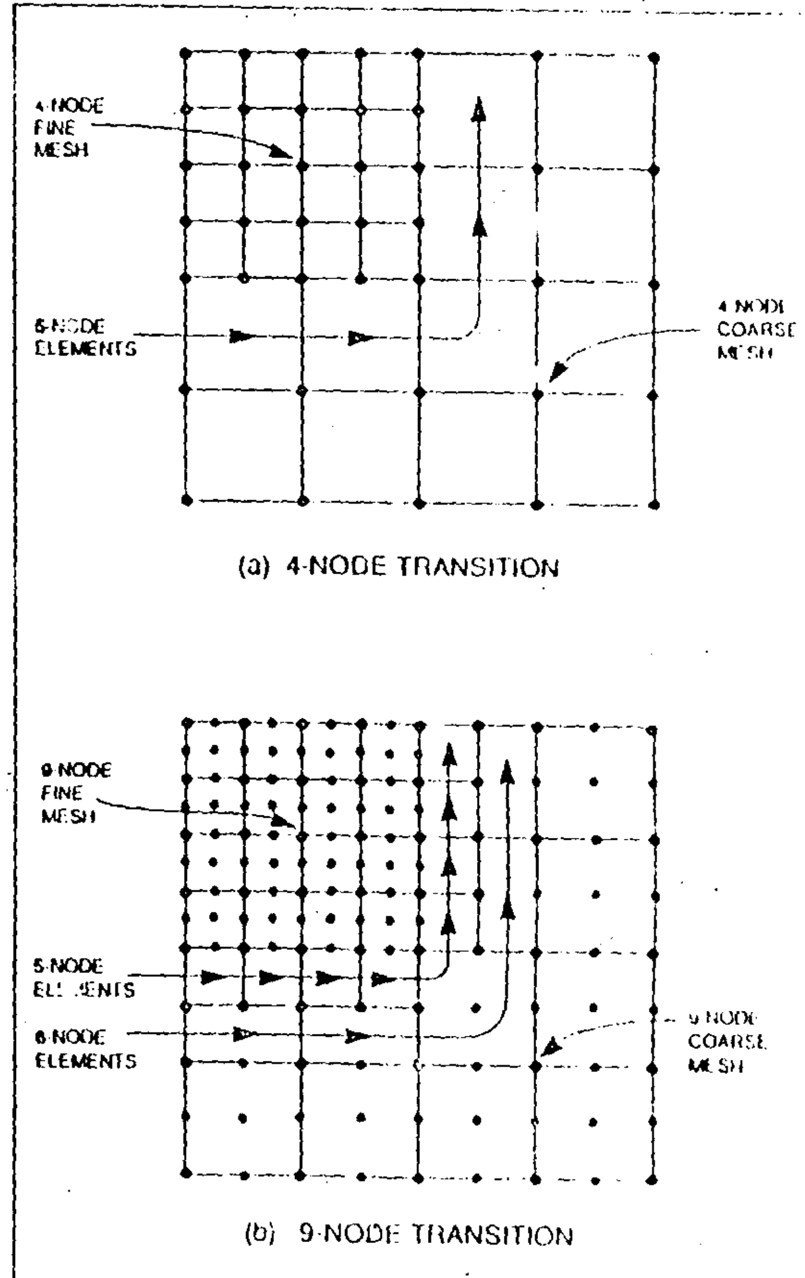


Figure IV-3
ASOLID Element Mesh Transitions

from different elements attached to a common joint. This will give an indication of the accuracy of the finite element approximation and can then be used as the basis for the selection of a new and more accurate finite element mesh.

The 9 node option is the recommended formulation.

D.4. The SOLID Element

This element is for modeling:
  Three-dimensional solid structures.

This is an 8-node brick element based upon an isoparametric formulation including nine optional incompatible bending modes.

Temperature-dependent, anisotropic material properties are allowed.

Element loading in the form of pressure gradients, thermal and gravity loads is allowed.

Element generation options are available.

A 2 x 2 x 2 numerical integration scheme is used for the elements. All stress values are calculated at the element joints in the global coordinate system.

The incompatible bending modes should be suppressed for distorted (nonrectangular) elements.

E. Rigid-Floor Diaphragm Modeling

The in-plane stiffness of most concrete floors (or concrete-filled decks) in building structures is, in general, very high.

SAP90 has a special option for modeling such horizontal rigid-floor diaphragm systems.

A floor diaphragm is modeled as a rigid horizontal plane parallel to the global X-Y plane, so that all points on any one floor diaphragm (at any one level) cannot displace relative to each other in the X-Y plane. See Figure IV-4.

Typically, each floor diaphragm is established by a joint in the plane of the diaphragm called the **master joint** of the diaphragm. The location of the master joint on each floor diaphragm is arbitrary and is selected by the user.

All of the other joints that exist on the diaphragm are connected to the master node by rigid links, and their displacements are dependent upon the displacements of the master joint. These joints are called **dependent** joints, and their displacements are defined by the following relationships which are automatically enforced by the program:

$$UX_d = UX_m + C_y\,RZ_m$$

$$UY_d = UY_m + C_x\,RZ_m$$

$$RZ_d = RZ_m$$

where:

$$C_x = X_d - X_m$$

$$C_y = Y_m - Y_d$$

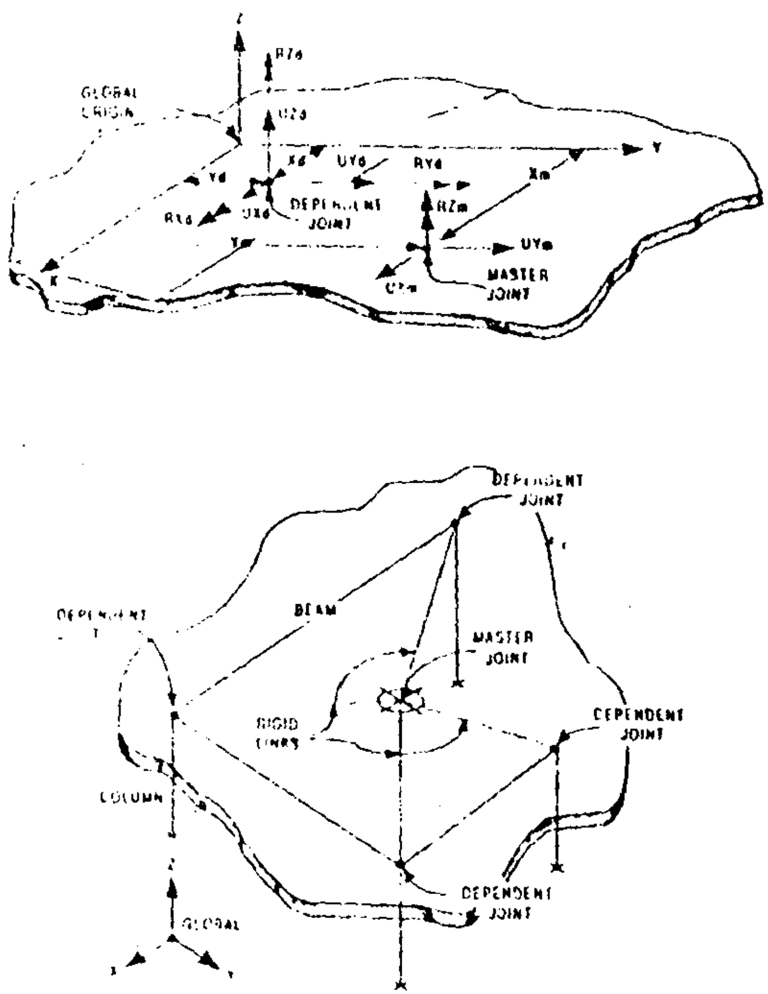


Figure IV-4
The Rigid Diaphragm Option

and UX_d, UY_d and RZ_d are the X-translation, Y-translation and Z-rotation, respectively, of a dependent joint; and UX_m, UY_m and RZ_m are the same, respectively, of the master joint.

This option effects a significant reduction in the number of equations which must be solved, thereby increasing the size of problem which can be accommodated on a microcomputer. Also, the rigid diaphragm option eliminates the numerical accuracy problems created when the large in-plane stiffness of a floor diaphragm is modeled with membrane elements.

Only FRAME elements may be connected to dependent joints. Also, any masses or loads at the joints that are assigned along the UX, UY or RZ degrees of freedom of the dependent joints will be lost.

This option is very useful in the lateral (X or Y direction) dynamic analysis of building type structures. Lumping the story masses at the center of mass (with an associated mass moment of inertia about the Z-axis) will result in a very small eigenvalue problem. See Figure IV-5 for mass moment of inertia formulations for various diaphragm configurations.

F. Pressure Gradient Loading

Structures subject to fluid loads, with fluid seepage, pore water pressures, pressure gradients and resulting buoyancy effects represent an area of analytical concern.

SAP90 has an option whereby the user can define the pressure distribution over the volume of a structure by assigning scalar pressure values to the joints of the model. These pressure values

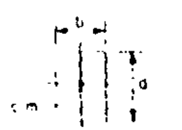
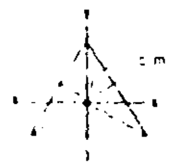
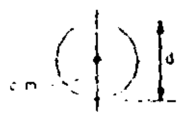
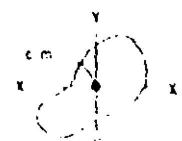
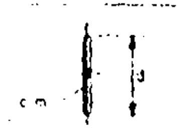
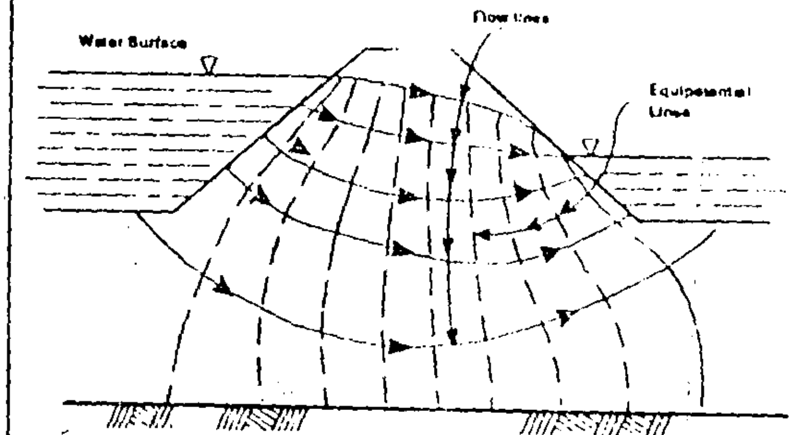
Shape in plan	Mass Moment of Inertia about vertical axis (normal to paper) through center of mass	Formula
	Rectangular diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	$MMI_{cm} = \frac{M(b^2 + d^2)}{12}$
	Triangular diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	Use general diaphragm formula
	Circular diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	$MMI_{cm} = \frac{Md^2}{8}$
	General diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g) Area of diaphragm = A Moment of inertia of area about XX = I_x Moment of inertia of area about YY = I_y	$MMI_{cm} = \frac{M(I_x + I_y)}{A}$
	Line mass Uniformly distributed mass per unit length Total mass of line = M (or w/g)	$MMI_{cm} = \frac{Md^2}{12}$
	Axis transformation for a mass If mass is a point mass, $MMI_o = 0$	$MMI_{cm} = MMI_o + MD^2$

Figure IV-5
Formulae for Mass Moment of Inertia



Earth Dam Flownet

Evaluation of Pore Water Pressure Distribution

Figure IV-6
Pressure Gradient Loading

may typically be obtained from a flownet analysis, such as is illustrated in Figure IV-6.

Based upon the elements that connect to the joints and the corresponding joint pressures, the program will evaluate three-dimensional element pressure load vectors to be assembled into the global load vector. The direction of such loads will obviously be directed from regions of high pressure values toward regions of low pressure values, resulting in an automatic evaluation of the overturning and uplift forces on the structure. See the POTENTIAL Data Block in Chapter X.

G. Graphics

The SAPLOT module is used for obtaining screen and hard-copy graphic displays of models that have been set up for analysis with SAP90. See Reference [5].

The program has options for plotting two-dimensional and three dimensional views displaying any of the following:

- Undeformed structural geometry
- Loads acting on the structure
- Static analysis deformed shapes
- Steady-state analysis deformed shapes
- Mode shapes
- FRAME element axial force, torsion, bending moment and shear diagrams
- SHELL, ASOLID, and SOLID element stress contours

The model can be viewed from any arbitrary direction. The user locates an arbitrary point with respect to the SAP90 global X-Y-Z coordinate system. This point is called the **view control point**. The view is set in the direction looking from the view control point towards the SAP90 global origin. This convention only sets the view direction. The actual location of the viewer's eye is assumed to be at infinity; in other words, the view is an infinite projection onto the plot plane, and the vector from the

view control point to the SAP90 global origin is the normal to the plot plane.

The SAP90 positive global X, Y and Z axes are plotted to show the orientation of the plot.

While displaying the deformed shape of a structure, the user may also plot the undeformed shape with dashed lines (as a "reference structure") allowing convenient comparison with the deformed shape. The user can set translational and rotational maxima to accentuate the structural deformation patterns. The deformed shapes of elements may be plotted with displaced straight lines or with bent cubic curves that preserve joint rotational compatibility at the ends of the members.

The program has an option whereby all elements may be shrunk about their centroids, thereby clearly displaying element connectivities and uncovering overlapped element boundaries.

All scaling of the views is automatic and the program has options to display "blowups" of localized regions of the structure.

Labeling options for identifying the joints and the elements are also available.

The SAPTIME module is used for obtaining screen and hard-copy graphic displays of input and response time functions for time history analysis done using SAP90. See Reference [6].

The program has options for plotting:

- Input time functions,

- Input base displacement, velocity, and acceleration time histories.
- Nodal displacement, velocity and acceleration time histories.

V.

STATIC AND DYNAMIC ANALYSIS

The following is a brief description of the static and dynamic analysis options of SAP90.

A. Static Analysis

The static analysis of a structure involves the solution of the system of linear equations represented by:

$$\mathbf{K}\,\mathbf{U} = \mathbf{R}$$

where **K** is the stiffness matrix,

U is the vector of resulting displacements,

and **R** is the vector of applied loads.

See Reference [17].

The structure may be analyzed for more than one load condition in a single run.

The static loading on the joints may be in the form of concentrated nodal loads or moments.

The static loads on the elements may take the following forms:

- For the FRAME elements:
  - Gravity loading
  - Span uniform loading
  - Span point loads
  - Span trapezoidal loading
  - Thermal loading, including thermal gradients through element depth and width
  - Prestress loading, from post-tensioning cables
- For the SHELL elements:
  - Gravity loading
  - Surface pressure loading
  - Thermal loading, with no thermal gradients through the shell thickness
- For the ASOLID and SOLID elements:
  - Gravity loading
  - Pressure gradient loading
  - Thermal loading

The steady-state analysis option may not be active in a static analysis run; however, an eigenvalue analysis and a response-spectrum analysis or a time-history analysis may be performed simultaneously with a static analysis in the same run.

B. Dynamic Analysis

The dynamic analysis options of SAP90 include the following:

- Steady-State Analysis
- Eigenvalue Analysis
- Ritz-Vector Analysis
- Response-Spectrum (Seismic) Analysis
- Time-History Analysis

These are described in detail below.

B.1. Steady-State Analysis

A very common type of loading is of the form $\mathbf{R} = \sin(\omega t)\mathbf{F}$, where ω is the circular frequency of the excitation, so that $\mathbf{R}$ varies with respect to time; however, the spatial distribution of load $\mathbf{F}$ does not vary as a function of time. For the case of zero damping, the equilibrium equations for the structural system are of the following form:

$$\mathbf{K\,U} + \mathbf{M\,W} = \mathbf{R} = \sin(\omega t)\,\mathbf{F}$$

where **K** is the stiffness matrix and **M** is the diagonal mass matrix. The steady state solution of this equation requires that the joint displacements **U** and accelerations **W** are of the following form:

$$\mathbf{U} = \sin(\omega t)\ \mathbf{A}$$

$$\mathbf{W} = -\omega^2 \sin(\omega t)\ \mathbf{A}$$

Therefore, the response amplitude **A** is given by the solution of the following set of linear equations:

$$[\mathbf{K} - \omega^2\ \mathbf{M}]\ \mathbf{A} = \mathbf{F}$$

It is of interest to note that the solution for static loads is nothing more than a solution of this equation with zero frequency. The loading frequency is specified in cycles per second.

The displacements printed by the program are the values of **A** (the maximum displacements) which vary as sin(ωt).

The member forces printed by the program are also maximum values, which vary as sin(ωt).

The following are the limitations of the steady-state analysis option:

- The structural damping is assumed to be zero.

- In any one run, the structure may be analyzed for more than one spatial distribution of steady state loads. However, the excitation frequency of *all* the loads in any one run must be the same.

- If the excitation frequency corresponds to a natural frequency of the structure, the system will go into resonance, resulting in an infinite response. In such cases the solution of the system will fail as the matrix $[\mathbf{K} - \omega^2\mathbf{M}]$ will be singular. The solver will report that the structure is unstable.

- No static analysis, eigenvalue analysis, Ritz-vector analysis, response-spectrum analysis or time-history analysis may be performed in a steady-state analysis run.

- A P-Delta analysis (Chapter VI) will generally be meaningless in a steady-state analysis run.

B.2. Eigenvalue Analysis

Seismic analysis by the response-spectrum approach or time-history analysis using the mode-superposition method requires determination of the undamped free-vibration mode shapes and frequencies of the system.

This involves the solution of the generalized eigenvalue problem

$$\mathbf{K}\ \Phi = \mathbf{M}\ \Phi\ \Omega^2$$

where: **K** is the stiffness matrix

M is the diagonal mass matrix

Ω^2 is the diagonal matrix of eigenvalues

Φ is the matrix of corresponding eigenvectors

SAP90 solves this eigenproblem for the lowest nfq eigenvalues (and corresponding vectors) by using an "accelerated subspace iteration" algorithm. During the solution phase, the program prints the "approximate eigenvalues" after each iteration. As the eigenvectors converge they are removed from the subspace and new approximate vectors are introduced. For details of the algorithm, see Reference [26].

B.3. Ritz-Vector Analysis

Recent research has indicated that the exact free-vibration mode shapes are not the best basis for a mode-superposition dynamic analysis of structures subjected to dynamic loads. It has been demonstrated [27] that dynamic analyses based on an unique set of Ritz vectors yield more accurate results than the use of the same number of exact mode shapes.

The reason the Ritz vectors yield excellent results is that they are generated by taking into account the spatial distribution of the dynamic loading, whereas the direct use of the exact mode shapes neglects this very important information. The first Ritz vector is the displacement vector obtained from a static analysis using the spatial distribution of the dynamic load vector as input. The other vectors are generated from a recurrence relationship in which the mass matrix is multiplied by the previously obtained Ritz vector and used as the load vector for a static solution. Therefore, after the stiffness matrix is triangularized it is only necessary to statically solve for one load vector for each Ritz vector required.

A transformation using the joint displacements of the Ritz vectors is used to reduce the number of dynamic equilibrium equations. Standard eigensolution techniques are then used to ...malize this reduced set of equations.

This results in an extremely efficient algorithm. Also, the method automatically includes the advantages of the proven numerical techniques of static condensation, Guyan reduction and static correction due to higher-mode truncation.

The algorithm is detailed in reference [23].

The SAP90 program uses the spatial distribution of structural mass to obtain the starting Ritz vectors. The use of these vectors is recommended for all base-motion problems.

B.4. Response-Spectrum (Seismic) Analysis

The dynamic equilibrium equations associated with the response of a structure to ground motion are given by

$$\mathbf{K\,U + C\,V + M\,W = M\,W_g}$$

where: $\mathbf{K}$ is the stiffness matrix

$\mathbf{C}$ is the damping matrix

$\mathbf{M}$ is the diagonal mass matrix

$\mathbf{W_g}$ is the ground acceleration

and **U**, **V** and **W** are the structural displacements, velocities and accelerations, respectively.

SAP90 will solve this system of equations using the mode-superposition response-spectrum approach. See Reference [24]. The ground acceleration is input as a digitized response-spectrum curve of spectral acceleration versus time period.

The ground excitation can occur simultaneously in three directions, namely, any two mutually perpendicular directions in the X-Y plane, and in the Z-direction. To get the maximum displacements and member forces (or stresses), the modal responses associated with a particular direction of excitation are first calculated and then combined using the Complete Quadratic Combination technique (CQC). See Reference [25].

The total response is then calculated by summing the responses from the three directions by the Square Root of the Sum of the Squares (SRSS) method.

In modeling structures subjected to dynamic response-spectrum loads, the positive Z-axis must point up.

B.5. Time-History Dynamic Response Analysis

In general, the loading which acts on a structure is an arbitrary function of space and time. In matrix form, it can be written as $\mathbf{R}(s,t)$. For all types of loading $\mathbf{R}(s,t)$ can be written as a finite sum of a series of spatial loading vectors $\mathbf{F}_i(s)$ and time functions $T_i(t)$. Or in matrix form:

$$\mathbf{R}(s,t) = \sum_i \mathbf{F}_i(s) * T_i(t)$$

For the SAP90 program the spatial vectors $\mathbf{F}_i(s)$ can be defined as the static load conditions, or in the case of base accelerations, as a function of the mass matrix. The functions $T_i(t)$ can be arbitrary functions of time or periodic functions such as those produced by wind or sea wave loading.

With SAP90, the standard mode-superposition method of response analysis is used to solve the dynamic equilibrium equations of motion for the complete structure. The modes used can be the undamped free-vibration modes (eigenvectors) or the load-dependent Ritz vectors. In the case of earthquake or base acceleration, utilizing the Ritz vectors will always produce more accurate results than if the same number of eigenvectors is used. Since the Ritz-vector algorithm is several times faster than the eigenvector algorithm, the former is recommended for base-acceleration types of input.

SAP90 performs "exact integration" of the modal-response equations for a linear variation of the time-function $T_i(t)$ between the input data time points. Therefore, the results are not dependent on the selection of a "time-integration interval" as in some other methods.

VI.

P-DELTA ANALYSIS

The SAP90 P-Delta analysis option permits consideration of the effect of an axial load upon the transverse bending behavior of FRAME elements. This is a type of geometric nonlinearity known as the P-Delta effect. This option is particularly useful for considering the effect of gravity loads upon the lateral stiffness of building structures, as required by certain design codes [14,15]. It can also be used for the analysis of some cable structures, such as suspension bridges, cable-stayed bridges, and guyed towers. Many other applications are possible.

This chapter describes the basic concepts behind the P-Delta effect and the implementation of the P-Delta option in SAP90. Details of the input data required are given in Chapter X. Modeling examples and guidelines can be found in the Verification Manual [8].

A. Types of Nonlinear Behavior

The P-Delta effect is only one of several different types of nonlinear structural behavior. These will be described in this section.

When the load acting on a structure and the resulting deflections are small enough, the load-deflection relationship for the structure is linear. For the most part, SAP90 analyses assume such

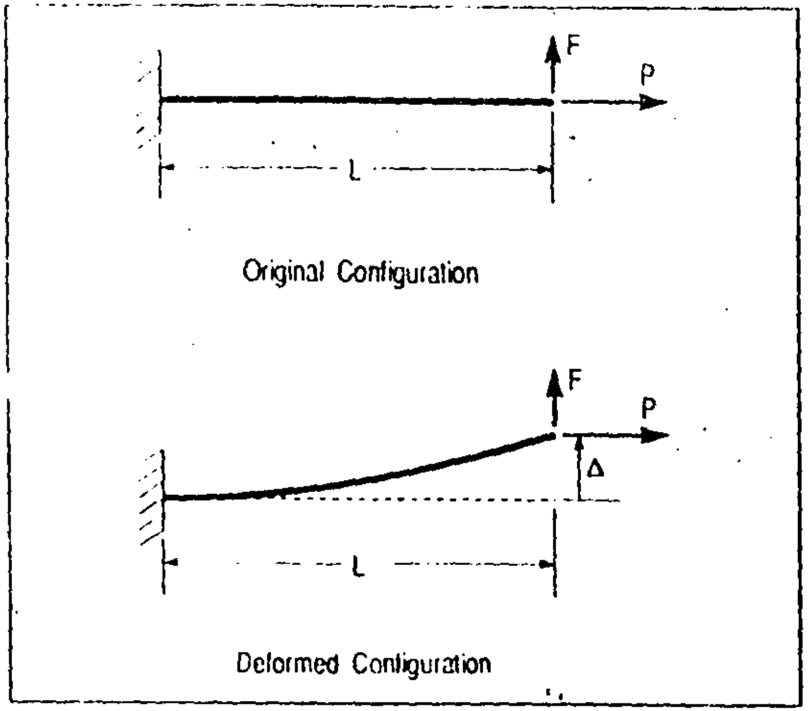


Figure VI-1
Cantilever Beam Example — Geometry

equilibrium is considered in the deformed configuration, there is an additional moment caused by the axial force P acting on the transverse tip displacement Δ. The moment no longer varies linearly along the length; the variation depends instead upon the deflected shape. The moment at the base is now $M = FL - P\Delta$. The moment diagrams for various cases are shown in Figure VI-2.

Note that only the transverse deflection is considered in the deformed configuration. Any change in moment due to a change in length of the member is neglected here.

If the beam is in tension, the moment at the base and throughout the member is reduced, hence the transverse bending deflection

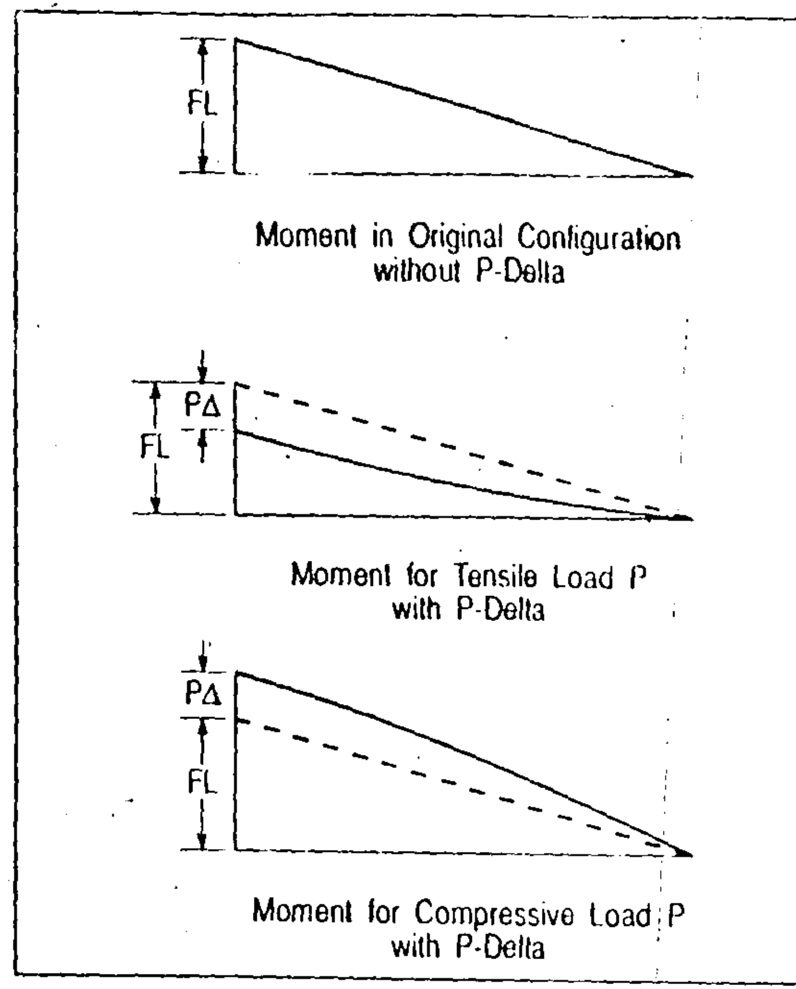


Figure VI-2
Cantilever Beam Example — Moment Diagrams

Δ is also reduced. Thus the member is effectively stiffer against the transverse load F.

Conversely, if the beam is in compression, the moment throughout the member, and hence the transverse bending deflection Δ, are now increased. The member is effectively more flexible against the load F.

linear behavior. This permits the program to form the equilibrium equations using the original (undeformed) geometry of the structure. Strictly speaking, the equilibrium equations should refer to the geometry of the structure after deformation.

The linear equilibrium equations are independent of the applied load and the resulting deflection. Thus the results of different static and/or dynamic load conditions can be superposed (scaled and added), resulting in great computational efficiency.

If the load on the structure and/or the resulting deflections are large, then the load-deflection behavior may become nonlinear. Several causes of this nonlinear behavior can be identified:

- **Large-stress effect:** when large stresses (or forces and moments) are present within a structure, equilibrium equations written for the original and the deformed geometries may differ significantly, even if the deformations are very small. The P-Delta effect is an example of this large stress effect.

- **Large-displacement effect:** when a structure undergoes large deformation (in particular, large strains and rotations), the usual engineering stress and strain measures no longer apply, and the equilibrium equations must be written for the deformed geometry. This is true even if the stresses are small.

- **Material nonlinearity:** when a material is strained beyond its proportional limit, the stress-strain relationship is no longer linear. Plastic materials strained beyond the yield point may exhibit history-dependent behavior. Material nonlinearity may affect the load-deflection behavior of a structure even when the equilibrium equations for the original geometry are still valid.

The large-stress and large-displacement effects are both termed kinematic (or geometric) nonlinearity, as distinguished from material nonlinearity. Kinematic nonlinearity may also be referred to as second-order geometric effects.

Other sources of nonlinearity are also possible, including nonlinear loads, boundary conditions and constraints.

The SAP90 P-Delta analysis option can be used to analyze the P-Delta effect, one type of large-stress effect. Other types of nonlinear behavior are not considered by the program.

B. The P-Delta Effect

The **P-Delta effect,** in this manual, refers specifically to the nonlinear effect of a tensile or compressive direct stress upon transverse bending and shear behavior. A compressive stress tends to make a structural member more flexible in transverse bending and shear, whereas a tensile stress tends to stiffen the member against transverse deformation.

The basic concepts behind the P-Delta effect are illustrated in the following example.

B.1. Cantilever Beam Example

Consider a cantilever beam subject to an axial load P and a transverse tip load F as shown in Figure VI-1. The internal axial force throughout the member is also equal to P.

If equilibrium is examined in the original configuration (using the undeformed geometry), the moment at the base is $M = FL$ and decreases linearly to zero at the loaded end. If, instead

If the compressive force is large enough, the transverse stiffness goes to zero and hence the deflection Δ tends to infinity; the structure is said to have buckled. The theoretical value of P at which this occurs is called the Euler buckling load for the beam; it is denoted by P_{cr} and is given by the formula

$$P_{cr} = \frac{\pi^2 EI}{4L^2}$$

where EI is the bending stiffness of the beam section.

The exact P-Delta effect of the axial load upon the transverse deflection and stiffness is a rather complicated function of the ratio of the force P to the buckling load P_{cr}. The true deflected shape of the beam, and hence the effect upon the moment diagram, is described by cubic functions under zero axial load, hyperbolic functions under tension, and trigonometric functions under compression.

B.2. P-Delta Effect in Other Structural Systems

The P-Delta effect can be present in any other beam configuration, such as simply-supported, fixed-fixed, etc. Other member types, such as plates, shells, and solids may also be affected. The P-Delta effect may apply locally to individual members, or globally to the structural system as a whole.

The key feature is that a large direct force in one direction, acting upon a small deflection in a perpendicular direction, produces a significant moment that affects the behavior of the member or structure. Direct force refers herein to tension or compression, rather than shear. If the deflection is small, then the moment produced is proportional to the deflection.

B.3. Nonlinear Analysis

If the axial force (or any direct force) in a member is constant and known *a priori*, as in the example above, then the equilibrium equations can be written directly in the deformed configuration. The same equilibrium equations could then be used for a variety of different loads, and the results superposed as for a linear structure, provided that the axial force remains unchanged.

In a more complicated structure, especially one that is redundant, the axial forces in the members may not be known *a priori*. A preliminary analysis must be performed to estimate the axial forces throughout the structure; the equilibrium equations can then be re-formed and re-solved taking these axial forces into account. This second analysis may produce different axial forces in the members if the changing stiffness causes a force redistribution. Additional iterations, each re-forming and re-solving the equilibrium equations, may be required, until the axial forces and the structural deflections converge, that is, until they do not significantly change from one analysis to the next.

Such an iterative type of solution is typical for nonlinear problems. Because each iteration requires forming and solving the equilibrium equations, the total solution time may be considerably longer than that required for a linear problem.

The final equilibrium equations obtained after convergence of the iteration process will be valid for any loading situation that does not change the axial forces in the members. The results for different loads can then be superposed as for a linear structure.

In general, however, the axial forces throughout a structure will depend upon the applied loads. In such a case, a separate iterative analysis may be required for each set of applied loads, and the results of these analyses *cannot* be superposed.

C. SAP90 Implementation

The SAP90 P-Delta analysis option provides considerable power and flexibility for analyzing the P-Delta effect in buildings, bridges and other structures. However, to make the most effective and efficient use of this capability, the user should fully understand the assumptions and restrictions that underlie the implementation of this feature in SAP90. These are described in detail in this section. Additional implications for modeling are discussed in Section D and in the Verification Manual [8].

The P-Delta effect is the only type of geometric nonlinearity modeled by SAP90. Specifically, SAP90 considers only the effect in FRAME elements of a large axial force upon small transverse deflections. Other types of large-stress effects are not usually important for structures, and are not considered. No large-displacement effects are considered.

C.1. Analysis Procedure

SAP90 obtains, by iterative analysis, the equilibrium equations that include the P-Delta effect due to a *single* set of applied loads. These equations are then used for *all* sets of loads that are to be applied to the structure. The results for these different loads may then be superposed.

Equilibrium Equations

The equilibrium equations are manifested in SAP90 in two ways:

(1) The element stiffness matrices, which are assembled (combined) to form the overall structure stiffness matrix;

(2) The local element stress-displacement relations, which are used to determine the internal element stresses (forces) for output.

The effective stiffness matrices and stress-transformation matrices are determined by the iterative procedure described later in this section.

Element Types

The P-Delta effect can be significant in any type of structure or solid object if the loads are large enough. However, at present, SAP90 considers the P-Delta effect only in FRAME elements.

The P-Delta option may be used with models containing any or all types of elements. Only the equilibrium equations, and hence the stiffnesses, associated with the FRAME elements will be affected by the presence of large axial forces. For SHELL, ASOLID and SOLID types of elements, the linear elastic stiffnesses will always be used, and the element stresses will not reflect the P-Delta effect within those elements.

FRAME elements should be used to model those portions of any structure that may carry the significant P-Delta loads.

P-Delta Load Combination

For each SAP90 model, as defined by a single input data file, a single static load combination may be defined that creates the P-Delta effect. This **P-Delta load combination** is an arbitrary combination of the usual SAP90 static load conditions. Each load condition may be scaled before it is added into this load combination. The load conditions may include joint forces and moments, self-weight, span loads, temperature and pressure loads.

Throughout this manual, this applied P-Delta load combination will be referred to as the **P-load**. The axial forces in the FRAME elements throughout the structure, induced by the P-load, will be referred to as the **P-forces**.

The P-load is independent of any of the load combinations defined in the COMBO or ENVELOPE data blocks. Forces from response spectrum, time-history and bridge moving-load analyses cannot be included in the P-load.

When the P-Delta option is activated, the structure is first subjected to the P-load and analyzed iteratively to determine the P-Delta effect upon the stiffness matrix and the FRAME element stress-displacement relations.

The resulting stiffness matrix and stress-displacement relations, including the P-Delta effect, are then used for all subsequent analyses. This includes the analyses for all static load conditions, combinations, envelopes, eigen and Ritz vectors, response-spectrum, time-history, bridge influence lines and bridge moving-load cases [12]. Because the same constant stiffness matrix is used, these analyses are all linear, resulting in great computational efficiency, and permitting superposition of the results.

Use of the P-Delta analysis option with steady-state analysis does not generally make sense, since this would imply that the P-load must oscillate with the frequency neye.

The acting P-force found in each FRAME element is printed in the .F3F file (see Chapter VIII). Displacements and other results due to the P-load are not printed out. If these are desired, a static load combination should be defined in the COMBO data block that is identical to the P-Delta load combination.

For physically consistent results, the axial load in the FRAME elements for any of these analyses should not differ significantly from the P-forces determined in the P-Delta analysis. If it does, the user must use engineering judgement to determine if the results are meaningful. Consideration should also be given to performing a consistent P-Delta analysis as described later.

Iterative Analysis Procedure

Starting with the initial elastic stiffness matrix (equilibrium equations) and with zero axial forces in the FRAME elements, the P-load is applied to the structure and the corresponding displacements and P-forces obtained. The stiffness matrix is then modified to account for the P-Delta effect of these axial forces, the P-load is again applied, and the corresponding displacements and P-forces obtained.

Each formation of the stiffness matrix, application of the P-load, and determination of the displacements and P-forces is called an **iteration**. Iterations may be repeated until the resulting displacements converge, that is, until they do not change significantly from one iteration to the next. This type of iteration is called "direct iteration" [16,19,28].

A relative displacement tolerance which measures convergence is specified in the input data file. If the relative change in displacement from one iteration to the next is less than the tolerance, then no further iterations are performed. The relative change in displacement is defined as the ratio of the maximum change in displacement to the largest displacement in either iteration. Note that rotational and translational displacements are treated equally.

The maximum number of iterations that the program is allowed to perform i: also specified. This is used to prevent excessive computational time, since each iteration requires about as much computational effort as a linear static analysis.

The initial iteration is termed the zero-th iteration. It is a standard linear analysis that is always performed whether or not the P-Delta option is used. The maximum num... of iterations limits the number of *additional* iterations performed that correct for the P-Delta effect. Setting the maximum number of iterations to zero turns off the P-Delta option.

If convergence has *not* been obtained after the maximum num... of iterations has been performed, then the results of the analysis may be meaningless, and they should be viewed with great skepticism. Failure to converge may be due to several causes:

- Too few iterations were permitted. A reasonable number is usually 2 to 5, although more may be required, depending on the particular problem at hand.

- A convergence tolerance that is too small is used. A reasonable value depends on the particular problem. Beware, however, that using a value that is ... may result in ...nvergence to meaningless results.

- The structure is near buckling. The struct... should be stiffened against buckling, or the magnitude of the P-load reduced.

Figure VI-3 shows a flow chart that describes the iteration process. See Chapter IX for more information on how this fits into the entire SAP90 execution procedure.

Dynamic Analyses

Eigenvectors and Ritz-vectors are obtained using the stiffness matrix as modified for the P-Delta effect. Compressive P-forces tend to soften the structure, lengthening the periods of vibration. Tensile P-forces tend to shorten the periods. In structures where the P-forces are both compressive and tensile, the effect on the periods is not easily predicted.

Response-spectrum analyses and time-history analyses are performed using these eigenvectors or Ritz vectors as a basis. Thus the static P-Delta effect is automatically included. Any additional P-Delta effect due to dynamic axial forces is not accounted for, since this would require many separate, nonlinear P-Delta analyses taking great computational effort.

Bridge Moving-Load Analyses

The SAP90 Bridge Analysis Module [12] computes influence lines and moving load response using the stiffness matrix as modified for the P-Delta effect. The additional P-Delta effect due to vehicle moving loads is not accounted for, since this would require many separate nonlinear P-Delta analyses taking great computational effort.

If the axial forces due to vehicle moving loads are significant in comparison to the dead-load axial forces, it may be necessary

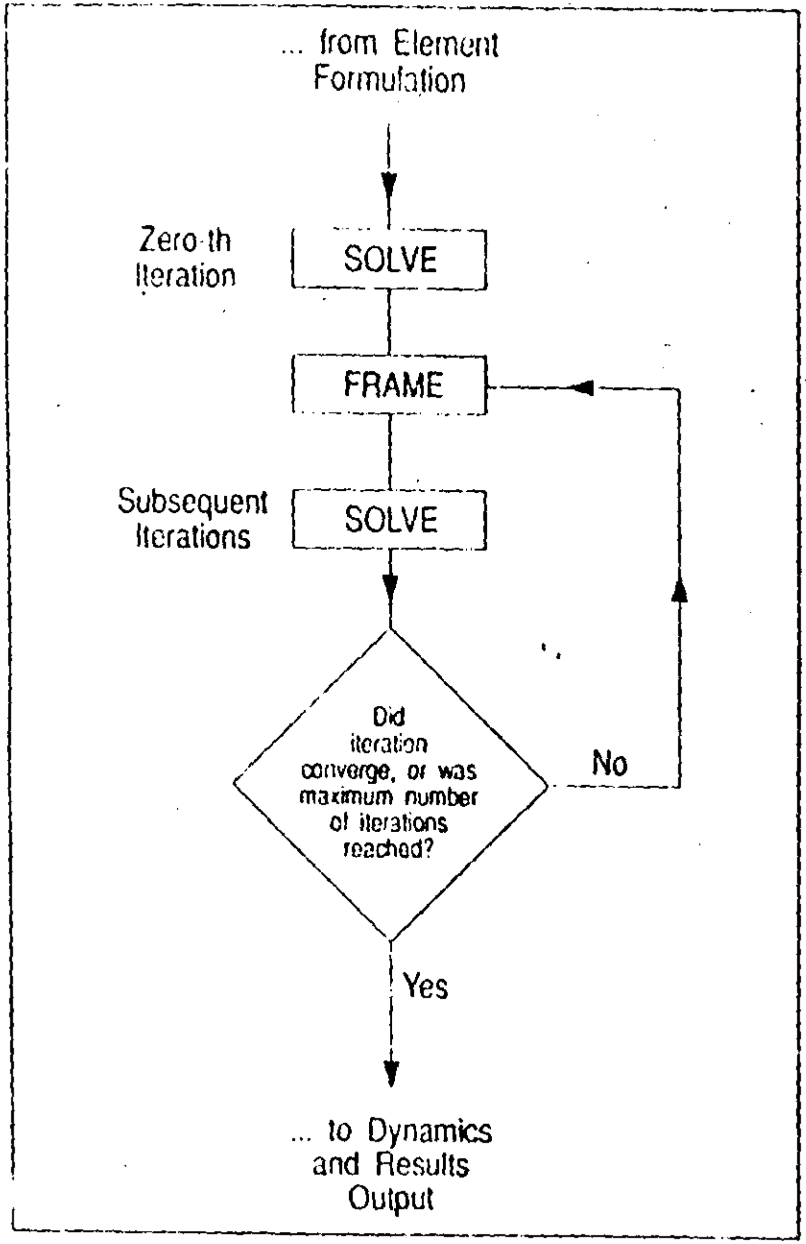


Figure VI-3
SAP90 Program Structure — P-Delta Iteration Process Only that Replaces SOLVE Module in Figure IX-2

to define the P-load so that it produces P-forces in the elements that include some average of the moving-load forces. This may require a prior SAP90 moving load analysis to determine these values.

Consistent P-Delta Analysis

To obtain P-Delta analyses that are more physically meaningful when the axial forces vary significantly from one load combination to the next, separate SAP90 analyses (separate input data files) can be used for each load combination. The load combination in the COMBO or ENVELOPE data block should be the same as that defining the P-load.

The results of these separate analyses cannot be superposed. The response to combined loads must be obtained by combining the loads themselves in a single run, not by combining the results from different runs.

It is not possible, in general, to have fully consistent dynamic or bridge moving-load analyses. However, useful and economical engineering results can usually be obtained with a judicious choice of the P-load.

C.2. FRAME Element

The implementation of the P-Delta effect in the FRAME element is described in the following.

Small Deflections

All deflections are assumed to be small. In particular, the magnitude of all strains and rotations are assumed to be much less than unity.

The P-Delta effect does not capture any change in *axial* stiffness of the member, such as might occur after buckling. This would require consideration of large-displacement effects.

Cubic Deflected Shape

The P-Delta effect is integrated along the length of each FRAME element, taking into account the deflection within the element. For this purpose the transverse deflected shape is assumed to be cubic for bending and linear for shear between the rigid zone offsets. The length of the rigid-zone offsets is reduced by the rigid-zone reduction factor.

The true deflected shape may differ somewhat from this assumed cubic/linear deflection in the following situations:

- The element has non-prismatic section properties. In this case the P-Delta deflected shape is computed as if the element were prismatic using the average of the properties from the two ends.
- Span loads are acting on the element (including self-weight, temperature and prestress). In this case the P-Delta deflected shape is computed using the equivalent fixed-end forces applied to the ends of the element.
- A large P-force is acting on the element. The true deflected shape is described by trigonometric functions under large compression, and by hyperbolic functions under large tension.

The assumed cubic shape is usually a good approximation to these shapes except under a compressive P-force near the buckling load with certain end restraints. Excellent results, however, can be obtained by dividing any structural member into two or more FRAME elements. See the Verification Manual [8] for more detail.

Note however that the design postprocessors SAPSTL [10] and SAPCON [11] expect that each structural member is modeled with a single FRAME element.

P-Delta Axial Forces

The P-Delta axial force (P-force) in each FRAME element is determined from the axial displacements computed in the previous iteration. *For meaningful results, it is important to use realistic values for the axial stiffness of these elements.* The axial stiffness is determined from the section properties that define the area and the modulus of elasticity. Using values that are too small may underestimate the P-Delta effect. Using values that are too large may make the P-Delta force in the element very sensitive to the iteration process.

Elements that have an axial force release, or that are constrained against axial deformation in the CONSTRAINT data block or as part of a rigid floor diaphragm, will have a zero P-force and hence no P-Delta effect.

The P-force is assumed to be constant over the length of each FRAME element. If the P-load includes axial span loads (or self-weight) that cause the axial force to vary, then the average axial force is used for computing the P-Delta effect. If the difference in axial force between the two ends of an element is small compared to the average axial force, then this approximation is usually reasonable. This would normally be the case for the columns in a building structure. If the difference is large, then the element should be divided into many smaller FRAME elements wherever the P-Delta effect is important. An example of the latter case could be a flagpole under self-weight.

Prestress

When prestress is applied to a FRAME element, the resulting P-force is assumed to be zero. This is because the tension in the internal prestressing cables is equal to the induced compression in the FRAME element representing the bending member; thus the two resulting P-Delta effects cancel. For this approach to be valid, it is necessary to assume that the prestressing cables act in narrow ducts within the bending member so that the transverse deflection of the bending member and the prestressing cables are the same.

If the prestressing cables are not constrained to move transversely with the bending member, and if P-Delta effects may be important, then the prestressing cables should be modeled as separate FRAME elements. The prestress force can be created in the prestressing cables by subjecting the elements to appropriate temperature decreases. P-Delta effects will then be computed separately for the prestressing cables and the bending member.

C.3. Buckling

If compressive P-forces are present and are large enough, the structure may buckle. Local buckling of individual members or global buckling of the whole structure are possible. SAP90 makes no distinction between local and global buckling.

If SAP90 detects that buckling has occurred, the analysis is terminated and no results are produced. This is because the analysis of a structure that has buckled requires consideration of large displacement effects that are not modeled by SAP90.

Detection of Buckling

Buckling may be detected during any iteration at one of two possible stages:

- During the formation of the stiffness matrix: if a sufficiently large, compressive P-force is present in a FRAME element, a negative value may be created on the diagonal of the stiffness matrix.

- During the solution of the equilibrium equations: a zero or negative value may be produced on the diagonal of the stiffness matrix.

In either case, the analysis will be terminated immediately.

Estimating the Buckling Load

SAP90 does not provide a direct method of determining the buckling load of a structure. It may be estimated, however, by performing a series of runs, each time increasing the magnitude of the P-load, until buckling is detected. It is important to keep unchanged the *relative* contributions from each load condition to the P-load, increasing all load condition scale factors by the same amount between runs.

It is important to understand that there is no single buckling load for a structure. Rather, there is a different buckling load corresponding to each spatial distribution of loads. If buckling of the structure is a concern under various loading situations, the buckling load should be estimated separately for each situation, as described above, by starting with different P-loads.

Local Buckling

The buckling of individual members within a structure can be detected provided that they are adequately modeled. For some support c... ations, a single FRAME element may adequately represent a structural member. However, for best results for all support conditions, two FRAME elements should be used to represent any structural member for which buckling may be a problem. The improvement obtained by using more than two elements does not usually warrant the additional effort.

D. Practical Application

This section provides some additional guidelines regarding practical use of the P-Delta analysis option. See also th. Verification Manual for example problems.

D.1. Preliminary Linear Analysis

After the structural model is developed, including the basic load conditions it is *strongly recommended* that a linear, static analysi... ...formed before running any P-Delta analyses. The results of this preliminary static analysis should be used to check the model for correctness before incorporating the complexity of the nonlinear P-Delta effect.

Examination of the deflected shape and the element forces and stresses will help detect errors in geometry, element c... tivity, material properties, restraints and constraints.

It is useful to define a static load combination in the COMBO data block equivalent to the P-Delta load co...oination. The FRAM.. element axial force distribution due to this load combination should be examined to make sure that the axial stiffnesses are reasonable, and that the axial force does not vary too much within any single FRAME element.

D.2. Building Structures

For most building structures, especially tall buildings, the P-Delta effect of most concern occurs in the columns due to gravity load, including dead and live load. The column axial forces are compressive, making the structure more flexible against lateral loads.

Building codes [14,15] normally recognize two types of P-Delta effects: the first due to the overall sway of the structure and the second due to the deformation of the member between its ends. The former effect is often significant; it can be accounted for fairly accurately by considering the total vertical load at a story level, which is due to gravity loads and is unaffected by any lateral loads. The latter effect is significant only in very slender columns or columns bent in single curvature (not the usual case); this requires consideration of axial forces in the members due to both gravity and lateral loads.

SAP90 can analyze both of these P-Delta effects. However, it is recommended that the former effect be accounted for in the SAP90 analysis, and the latter effect be accounted for in design by using the applicable building-code moment-magnification factors [22]. This is how the SAP90 postprocessors -- SAPSTL [10] for steel frame design and SAPCON [11] for concrete frame design—are set up.

The P-Delta effect due to the sway of the structure can be accounted for accurately and efficiently, even if each column is modeled by a single FRAME element, by using the factored dead and live loads as the P-load. All static and dynamic load conditions utilize this same P-load. The iterative P-Delta analysis should converge rapidly, usually requiring only a single iteration.

As an example, suppose that the building code requires the following load combinations to be considered for design:

(1) 1.4 dead load

(2) 1.2 dead + 1.6 live load

(3) 1.2 dead + 0.5 live + 1.3 wind load

(4) 1.2 dead + 0.5 live – 1.3 wind load

(5) 0.9 dead + 1.3 wind load

(6) 0.9 dead + 1.3 wind load

For this case, the P-Delta effect due to overall sway of the structure can usually be accounted for, conservatively, by specifying the P-load to be 1.2 times the dead load plus 0.5 times the live load. This will accurately account for this effect in load combinations 3 and 4 above, and will conservatively account for this effect in load combinations 5 and 6. This P-Delta effect is not generally important in load combinations 1 and 2 since there is no lateral load.

The P-Delta effect due to the deformation of the member between its ends can be accurately analyzed only when separate SAP90 runs are made for each load combination, with the P-load specified equal to the combination being analyzed. Six runs would be needed for the example above. Also, at least two FRAME elements per column should be used. Again, it is

recommended that this effect be accounted for instead by using SAPSTL or SAPCON.

D.3. Cable Structures

The P-Delta effect can be a very important contributor to the stiffness of suspension bridges, cable-stayed bridges, and other cable structures. The lateral stiffness of cables is due almost entirely to tension, since they are very flexible in bending when unstressed. SAP90 can model this cable behavior provided that the cable geometry does not change too much upon loading.

In many cable structures, the tension in the cables is due primarily to gravity load, and it is relatively unaffected by other loads. If this is the case, it is appropriate to define the P-load to be a realistic combination of the dead load and live load.

It is usually important to use realistic values for the P-load, since the lateral stiffness of the cables is approximately proportional to the P-forces.

FRAME elements are used to model cables. A single element is sufficient between points of concentrated load. Additional elements may be needed if significant distributed loads, including self weight, act upon the cable. Concentrated loads should only be applied at joints, not as FRAME element span loads, since cables "kink" at such loads.

Each FRAME cable element should be given a small, realistic bending stiffness. Otherwise the structure may be unstable in the zero-th iteration before the tensile P-forces can provide lateral stiffness. For the same reason, moment end-releases should generally not be used for cable elements.

The geometry of a loaded cable is strongly dependent upon the type of loading applied. Because the SAP90 only considers small deflections, it is important to define the cable geometry (joint coordinates) to be close to what is expected after the structure is loaded. It may be necessary to correct the geometry after one or more preliminary runs that determine the shape of the cable under P-load. If the stretching of the cable under P-load is large (more than a few percent) it may not be possible to obtain meaningful results with the P-Delta option.

The P-Delta effect only affects transverse stiffness, not axial stiffness. Therefore, FRAME elements representing a cable can carry compression as well as tension; this type of behavior is generally unrealistic. The user should review the analysis results to make sure that this does not occur.

Because convergence tends to be slower for stiffening than softening structures, the nonlinear P-Delta analysis may require many iterations. Five to ten iterations would not be unusual.

D.4. Guyed Towers

In guyed towers and similar structures, the cables are under a large tension produced by mechanical methods that shorten the length of the cables. These structures can be analyzed by the same methods discussed above for cabled bridges.

The P-load in the guys can be defined to be a temperature decrease in the cables that will produce the requisite shortening. Several analyses may be required to determine the magnitude of the temperature change needed to produce the desired amount of cable tension.

VII. SAP90 INPUT DATA FILE STRUCTURE

A. Data Blocks

SAP90 input data is organized into twenty-one distinct data blocks by means of corresponding unique separator lines (except in the case of the Title Line data, where no separator exists). The separator identifies the data block and is always the first line in the data block. The separator line may be typed in uppercase or lowercase but it **must start in column 1.** Data associated with the data block immediately follows the separator line.

The twenty-one input data blocks are shown in Figure VII-1. Some of these data blocks are mandatory; however, the existence of most of the data blocks in the input data depends on the problem being analyzed. The order in which the data blocks occur in the input file is immaterial, however the Title Line should be the first line in the input file. The various functions of these data blocks and when they are needed are summarized in Figure VII-1.

Additional data blocks may be created by the preprocessor SAPIN [4] if it used to prepare the data. These are for use by SAPIN itself and are ignored by SAP90.

11 "ASOLID" Data Block

This data block defines the properties, locations and loadings associated with the three- to nine-node isoparametric element. This element can be used for modeling axisymmetric solids, plane-strain structures and plane-stress structures. All elements must ex[illegible]allel to the global principal planes, i.e. the X-Y, Y-Z or Z-X planes.

Skip this data block if there are no ASOLID elements in the model.

Otherwise, prepare data for Format Sections a through d as described below.

FORMAT

a. Separator

Provide one data line for the ASOLID separator in the following form:

ASOLID

b. ASOLID Control Information

Provide one data line for the ASOLID control information in the following form:

NM=nmat ETYPE=et MAXN=ntm
X=$x_1, x_2, \ldots, x_{nld}$ Y=$y_1, y_2, \ldots, y_{nld}$ Z=$z_1, z_2, \ldots, z_{nld}$
T=$t_1, t_2, \ldots, t_{nld}$ P=$p_1, p_2, \ldots, p_{nld}$ R=$r_1, r_2, \ldots, r_{nld}$

c. Material Property Data

Provide one set of data for each of the **nmat** material property types. Each data set co[illegible]sists of a first data line, immediately followed by temperature-dependent material property data as shown b[illegible].

(i) First Data Line

Prepare one data line in the following form:

nm NUMT=nt W=w M=m B=b

(ii) Temperature-Dependent Material Property Data

Prepare **nt** data lines in the following form:

T=t E=e_r, e_s, e_t U=u_1, u_2, u_3 G=g_{rs} A=a_r, a_s, a_t

d. ASOLID Element Location Data

In this data section, provide as many data lines as needed to define all the ASOLID elements in the model. End this [illegible]ata section with a blank line. Prepare the data lines in the following form:

nel JN=$j_1, j_2, j_3, j_4, \ldots j_9$ JQ=j_1, j_3, j_7, j_9 JS=j_i, j_j, j_k
M=mat TZ=tz TH=th G=g_1, g_2 LP=n

DESCRIPTION

Variable	Note	Default	Entry
Asolid Control Information			
nmat	(1)	[1]	Number of element material types
et	(2)	[0]	Element type =0 Axisymmetric =1 Plane-strain =2 Plane stress
ntm	(3)	[1]	Maximum number of temperatures in any one of the nmat material property sets
$x_1, x_2, \ldots$	(4)	[0]	X-direction gravitational multipliers for the **nld** structural load conditions
$y_1, y_2, \ldots$	(4)	[0]	Y-direction gravitational multipliers for the **nld** structural load conditions
$z_1, z_2, \ldots$	(4)	[0]	Z-direction gravitational multipliers for the **nld** structural load conditions
$t_1, t_2, \ldots$	(4)	[0]	Temperature multipliers for the **nld** structural load conditions
$p_1, p_2, \ldots$	(4)	[0]	Pressure multipliers for the **nld** structural load conditions
$r_1, r_2, \ldots$	(5)	[0]	Angular velocity values for the **nld** structural load conditions (rad/T units)

Variable	Note	Default	Entry
Material Property Data			
nm	(6)		Material identification number
nt	(7)	[1]	Number of temperatures for which material-dependent properties are specified for this material type
w	(8)	[0]	Weight per unit volume
m	(9)	[0]	Mass per unit volume
b	(10)	[0]	Material property reference angle (degrees). See Figure X-21
t	(11)	[0]	Temperature associated with material properties specified on this data line
e_r, e_s, e_t	(12)	[0]	Modulus of elasticity, in the r-, s- and t-directions, respectively
u_1, u_2, u_3	(12)	[0]	Poisson's ratio, as defined in the material law in Figure X-21
g_{rs}	(12)	$[\frac{e_r}{2+2u_1}]$	Shear modulus in the r-s plane
a_r, a_s, a_t	(12)	[0]	Coefficient of thermal expansion, in the r-, s- and t-directions, respectively (L/L/H units)

8. The entry **et** defines the type of element to be formulated. The options are:

et=0	For shell elements (with bending and membrane behavior)
et=1	For membrane elements
et=2	For plate bending elements

9. **mat** refers back to the material property table defined in Format Section c; **mat** must be a positive number not greater than **nmat**.

10. The joint temperatures specified in the POTENTIAL data block are used to induce thermal load in the elements. The element zero-stress reference temperature is subtracted from the element joint temperatures defined in the POTENTIAL data block to compute the temperature differences that produce the thermal strains. Therefore, if a structure is heated to 600 degrees and the zero stress reference temperature is 100 degrees, the thermal strains will be based upon a temperature increase of 500 degrees.

11. The element membrane thickness is used for calculating the element membrane stiffness, as well as the element volume for the element self-weight and mass calculation.

    The element bending thickness is used for calculating the element bending stiffness. If not specified it is taken equal to the element membrane thickness.

12. The user must clearly understand the definition of the element local 1-2-3 coordinate system in reference to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems.

The local Axis 3 is always the vector normal to the plane of the shell element. Axis 1 and Axis 2 are defined with the LP=n option, which activates the definition of a vector V_n as described below.

If n=1,	V_n is set parallel to the +X global axis.
If n=2,	V_n is set parallel to the +Y global axis.
If n=3,	V_n is set parallel to the +Z global axis.
If n=−1,	V_n is set parallel to the −X global axis.
If n=−2,	V_n is set parallel to the −Y global axis.
If n=−3,	V_n is set parallel to the −Z global axis.

Axis 1 and Axis 2 are then defined by the following cross products:

$$V_1 = V_n \times V_3$$
$$V_2 \quad V_3 \times V_1$$

The default case is defined as follows:

If n=0,	V_1 is the vector that is directed from the midpoint of the element edge I-K to the midpoint of the element edge J-L.

and V_2 is calculated as above.

Therefore, as illustrated in Figure X-18, if **n** equals 1 or −1, the V_1 vector is the line defined by the intersection of the element plane and the global Y-Z plane.

If **n** equals 2 or −2, the V_1 vector is the line defined by the intersection of the element plane and the global X-Z plane.

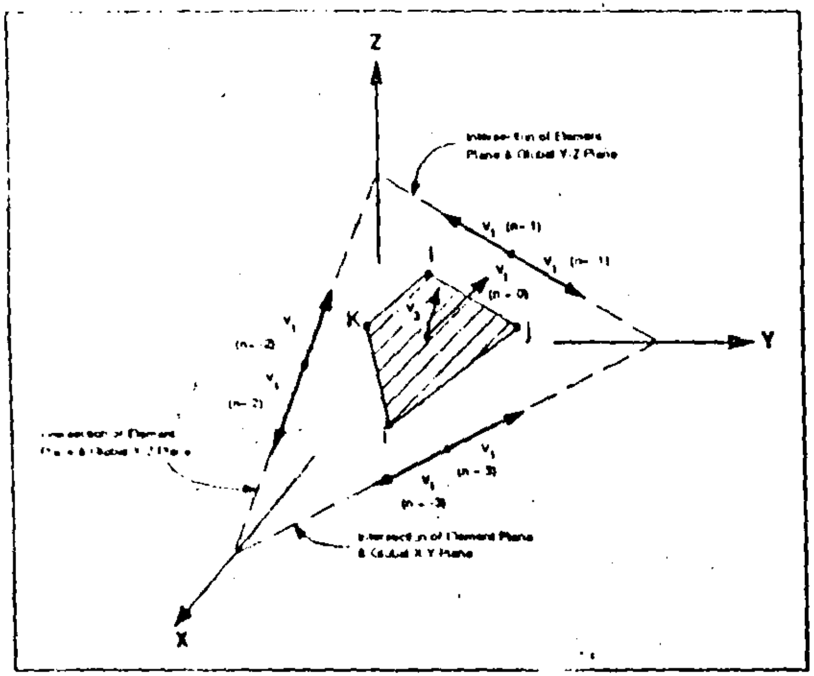


Figure X-18
Shell Element Local Axis

If n equals 3 or -3, the V_1 vector is the line defined by the intersection of the element plane and the global X-Y plane.

13. g_1 and g_2 are parameters that cause the generation of a two-dimensional mesh with g_1 elements in the I-J direction and g_2 elements in the I-K direction. The value of g_1 must not be less than 1, since this number includes the current element being defined; similarly for g_2. See Figure X-19.

The node numbers of the generated elements are formed by incrementing the node numbers of the basic element by $(j_J - j_I)$ in the I-J direction and by $(j_K - j_I)$ in the I-K direction. Therefore, generation is restricted to meshes with regular numbering systems. The element identification numbers for generated elements are obtained by incrementing the

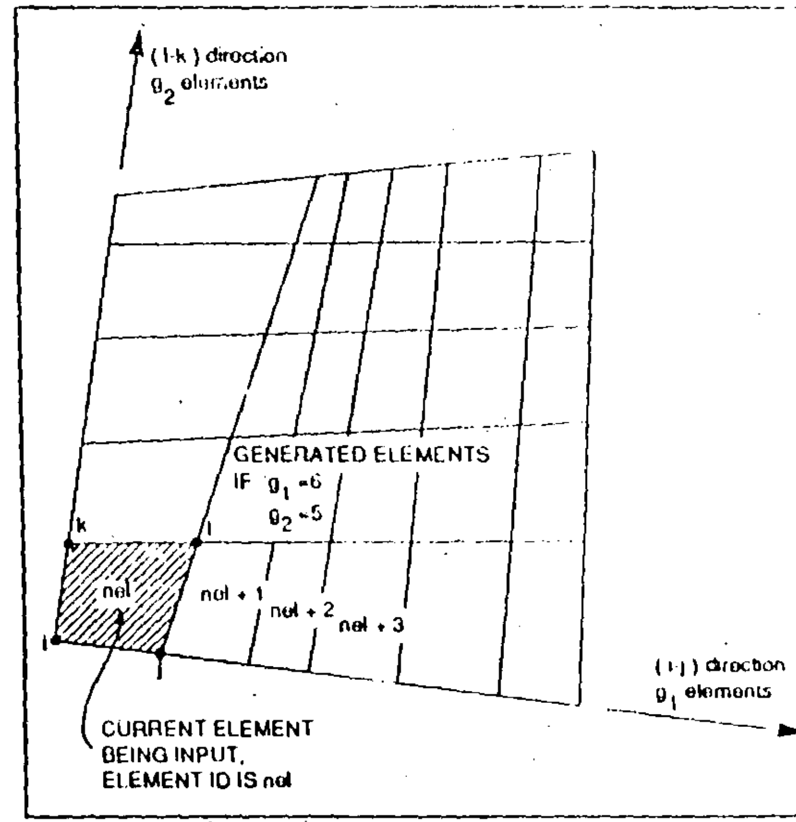


Figure X-19
Shell Element Generation

identification number of the previous element by 1. All o the generated elements are assigned the same materi properties, type, thickness, reference temperature and loca axis flags.

When deleting elements only g_1 may be specified, givin the *total* number of elements being deleted. The identifica tion numbers of the deleted elements are assumed to incre ment by 1. See Note 6.

Variable	Note	Default	Entry
Material Property Data			
nm	(4)		Material identification number
e		[pv]	Modulus of elasticity
u		[pv]	Poisson's ratio
w	(5)	[0]	Weight per unit volume
m	(5)	[0]	Mass per unit volume
alpha		[0]	Coefficient of thermal expansion (L/L/H units)
Shell Element Location Data			
nel	(6)		Element identification number
ji, jj,...	(7)		Element joint numbers
et	(8)	[pv]	Element type =0 Shell (membrane plus bending) =1 Membrane behavior only =2 Plate-bending behavior only
mat	(9)	[pv]	Element material type
tz	(10)	[0]	Zero-stress reference temperature
th_1	(11)	[pv]	Element membrane thickness
th_2	(11)	[pv]	Element bending thickness
n	(12)	[0]	Element local axis direction flag
g_1, g_2	(13)		Element generation parameters

NOTES

1. The control parameter **nm** defines the number of data lines the program expects to read in the material property data section (Format Section c).

2. Normally shell output is in the form of resultant moments and forces per unit in-plane length. The iout parameter can be used to obtain results in the form of stresses. This parameter also affects results obtainable through SAPLOT

3. The **nld** load multipliers associated with the X-direction ($x_1, x_2, \ldots, x_{nld}$) correspond to the **nld** structural load conditions. These are gravitational multipliers that activate the self weight of the SHELL elements in the X-direction. In other words, static loads acting in the X-direction equal to the self weight of the SHELL elements, factored by the gravitational multipliers, will be added to the corresponding load condition.

   For example, if $x_2 = 1.4$, static loads consisting of 1.4 times the weight of the SHELL elements, acting in the positive global X-direction, will be added to the structural load condition 2. Note that only elements that have nonzero weights per unit volume contribute to the static vectors. Similarly, the **nld** Y-direction multipliers generate self load vectors in the Y-direction and the **nld** Z-direction multipliers generate self load vectors in the Z-direction.

   The **nld** temperature multipliers ($t_1, t_2, \ldots, t_{nld}$) and pressure multipliers ($p_1, p_2, \ldots, p_{nld}$) similarly generate thermal and pressure body forces for the corresponding load condition using the joint temperature and pressure values specified in the POTENTIAL data block.

4. The material property identification numbers must be in ascending, consecutive, numerical sequence starting with the number one (1).

5. The weight per unit volume is used for calculating the self-weight of the element. The self-weight is added into the structural load conditions via the gravitational load multipliers described above in Note 2.

   The mass per unit volume is used for the calculation of the mass of the element. Consistent mass units must be used. This entry is only needed in a dynamic analysis mode for automatic lumping of the element mass to the element joints when assembling the structural mass matrix.

6. The element identification number can be any number between 1 and **nid** (SYSTEM data block). Elements numbers do not have to be consecutive and may be supplied in any order.

   Elements may be re-specified or re-generated, in which case only the last definition is used. When an element is redefined the previous definition is completely lost; all unspecified variables use the standard default values, and "previous-value" defaults refer to the previous data line, not to the previous definition of the element being redefined.

   A previously defined element can be deleted by setting **jid** to the *negative* of its identification number. This may be used, for example, to create gaps within regions of generated elements. The only other data permitted on the data line when deleting elements is G=**g1** which specifies the total number of elements to be deleted; the element identification numbers increment by 1. See Note 13.

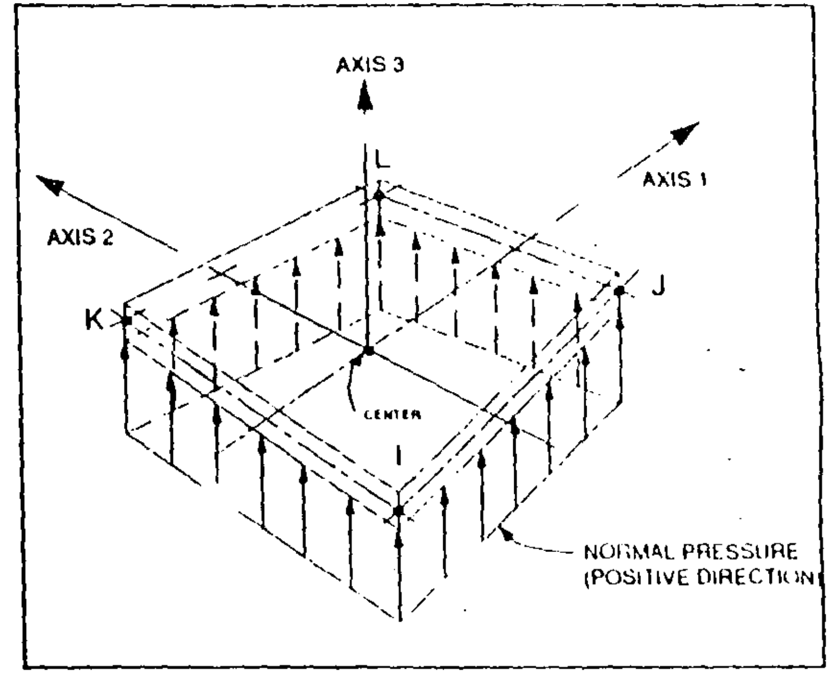


Figure X-17
Shell Element Connectivity and Pressure Loading
Three-dimensional Shell Element

7. **ji, jj, jk** and **jl** are the four joint numbers that describe the quadrilateral SHELL element. The four joints do not have to be coplanar. A small amount of twist in the element is accounted for by the program. These joint numbers must have been previously defined in the JOINTS data block. The sequence of input of the joint numbers that describe the element should be as shown in Figure X-17.

   A three-node triangular element may be specified as follows:

   JQ=**ji, jj, jk**

27. It has been found that an analysis based upon rigid offset lengths to the outer face of the supports can underestimate the deflections of the structure. The rigid zone reduction factor will reduce the lengths of the rigid offsets used, thereby compensating for some of the deformations that do exist in the zone bounded by the finite dimensions of the joint. The flexible element length L* is given by

$$L^* = L - (1 - z)(r_i + r_j)$$

A recommended value of z is 0.5. The clear length remains the same as defined in Note 26. Irrespective of the value of z, the moments and shear forces will always be output at the faces of the supports. See Note 3.

22

28. See Chapter IV, Section E.

29. The **nld** entries l_1, l_2,..., l_{nld} are references to the span loading patterns defined in Format Section d.

These entries define the manner in which the span loadings for this element are associated with the structural load conditions.

Each of the **nld** entries l_1, l_2,..., l_{nld} must be a non-negative number not greater than **nbsl**. Thus if l_2 = 7, span loading type 7 will be applied to this member for load condition 2; if l_3 = 0, no span load will be applied for Load Condition 3.

30. The value of n_g does not include the current element being defined. The element identification number, joint number at End I, joint number at End J, master joint number for End I and master joint number for End J, for each successive element are obtained by incrementing the values associated

with the current element by none, g_1, g_2, g_3 and g_4, respectively.

All of the generated members will have the same properties, release codes, rigid end offsets, master-joint specifications and beam span loadings as defined by this data line.

When deleting elements, only n_g and nine may be specified. See Note 21.

10. "SHELL" Data Block

This data block defines the properties, locations and loadings associated with the general three-dimensional four-node SHELL element. Three-dimensional plate bending and membrane elements are considered as special cases of this general element. Skip this data block if there are no SHELL elements in the model. Otherwise prepare data for Format Sections a through d as described below.

FORMAT

a. Separator

Provide one data line for the SHELL separator in the following form:

SHELL

b. SHELL Control Information

Provide one data line for the SHELL control information in the following form:

NM=nmat O=iout
$X=x_1,x_2,\ldots,x_{nld}$ $Y=y_1,y_2,\ldots,y_{nld}$ $Z=z_1,z_2,\ldots,z_{nld}$
$T=t_1,t_2,\ldots,t_{nld}$ $P=p_1,p_2,\ldots,p_{nld}$

c. Material Property Data

Provide **nmat** data lines in this data section to define the **nmat** material types in the following form:

n E=e U=u W=w M=m TA=alpha

d. SHELL Element Location Data

In this data section provide as many data lines as needed to define all the SHELL elements in the model. **End this data section with a blank line.** Prepare the data lines in the following form:

nel $JQ=j_i, j_j, j_k, j_l$ ETYPE=et M=mat TZ=tz
$TH=th_1, th_2$ LP=n $G=g_1, g_2$

DESCRIPTION

Variable	Note	Default	Entry
Shell Control Information			
nmat	(1)	[1]	Number of material types
iout	(2)	[0]	Flag for shell output: =0 Force and moment resultants =1 Top and bottom surface stresses
$x_1,x_2,\ldots$	(3)	[0]	X-direction gravitational multipliers
$y_1,y_2,\ldots$		[0]	Y-direction gravitational multipliers
$z_1,z_2,\ldots$		[0]	Z-direction gravitational multipliers
$t_1,t_2,\ldots$		[0]	Temperature multipliers
$p_1,p_2,\ldots$		[0]	Pressure multipliers

***Method 2. Global Vector Specification,* LP=n1,0**

This option is activated if **n1** is specified and **n2** is zero, for which case **n1** may have the value 1, 2, 3, –1, –2 or –3. With this option, the vector V_n as shown in Figure X-14 is set parallel to one of the global axes.

If LP = 1,0	V_n is set parallel to the positive global Z-axis.
If LP = 2,0	V_n is set parallel to the positive global Y-axis.
If LP = 3,0	V_n is set parallel to the positive global X-axis.

Using LP= -1,0 or LP= –2,0 or LP= –3,0 would have a similar effect, except that V_n will be set parallel to the negative global Z-axis, Y-axis or X-axis, respectively.

Axis 2 and Axis 3 are then defined by the following cross products:

$$V_2 = V_n \times V_1$$
$$V_3 = V_1 \times V_2$$

This option is very convenient, of course, if Axis 3 is parallel to one of the global axes.

If LP = 1,0	Axis 3 is set parallel to the positive global Z-axis.
If LP = 2,0	Axis 3 is set parallel to the positive global Y-axis.
If LP = 3,0	Axis 3 is set parallel to the positive global X-axis.

Similarly for the negative specifications.

***Method 3. K-Joint Specification,* LP=0,k**

This option is activated if **n2** is specified and **n1** is zero, in which case **n2** is a joint number, **k**, defining the vector V_k directed from the element joint j_i to joint **k** as shown in Figure X-14.

The vector V_k lies in the local 1-2 plane. Axis 2 and Axis 3 are then defined by the following cross products:

$$V_3 = V_1 \times V_k$$
$$V_2 = V_3 \times V_1$$

If needed, additional joints, that do not necessarily connect to any elements, may be added to the model for defining the required vectors.

25. When more than one element connects to a joint and it is known that certain element forces at that joint of a particular element are zero, the release codes associated with those element forces of the element need to be activated. In the example shown in Figure X-15, the diagonal element has a moment connection at End I and a pin connection at End J. All the other elements connecting to the joint at End J are continuous. Therefore, in order to model the pin condition the Moment 3 at End J should be set to zero. This is achieved by defining the element release code set for the diagonal member as follows:

LR=0,1,0,0,0,0

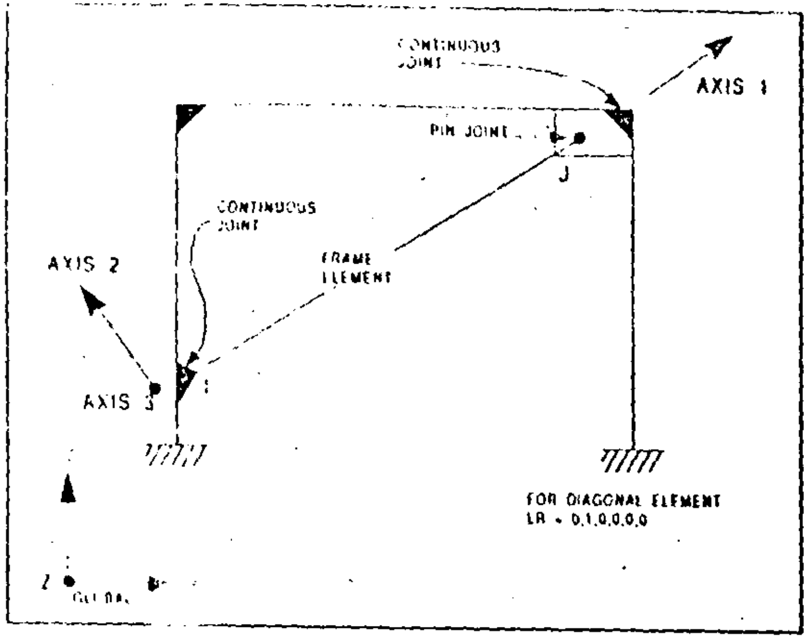


Figure X-15
Member End Release Codes

A one (1) value for any of the release options r_1 through r_6 activates the corresponding release condition. A zero (0) will retain continuity.

26. All structural members have finite cross-sectional dimensions. In many structures the dimensions of the members are large and have a significant effect on the stiffness of the structure. An analysis based upon a centerline-to-centerline geometry, in general, overestimates the deflections. Also, engineers prefer to have element forces output at the support faces.

The rigid offsets are the distances from the joints to the faces of the supports. See Figure X-16. There are no element bending and shear deformations within the rigid offset lengths and the moments and shear forces are output at the

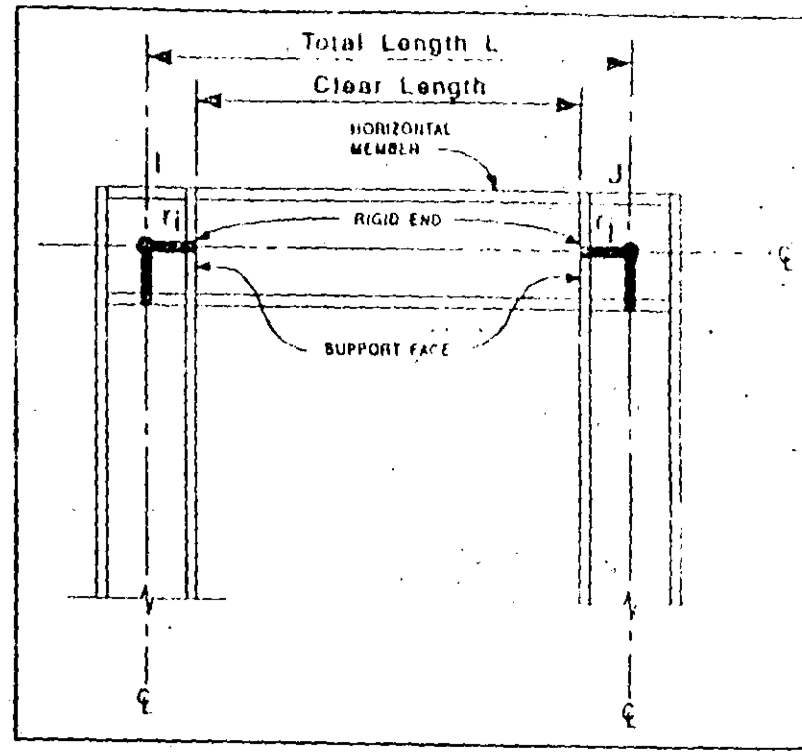


Figure X-16
Rigid End Offsets

outer ends of the rigid offsets, i.e., at the faces of the supports; see Note 3. The flexible length L* of the element is given by

$$L^* = L - (r_i + r_j)$$

where L is the total element length. The clear length of the element (between support faces) is the same as the flexible length.

If moment release is specified together with rigid end offsets, the moment hinge is modeled by the program at the end of the clear length of the element.

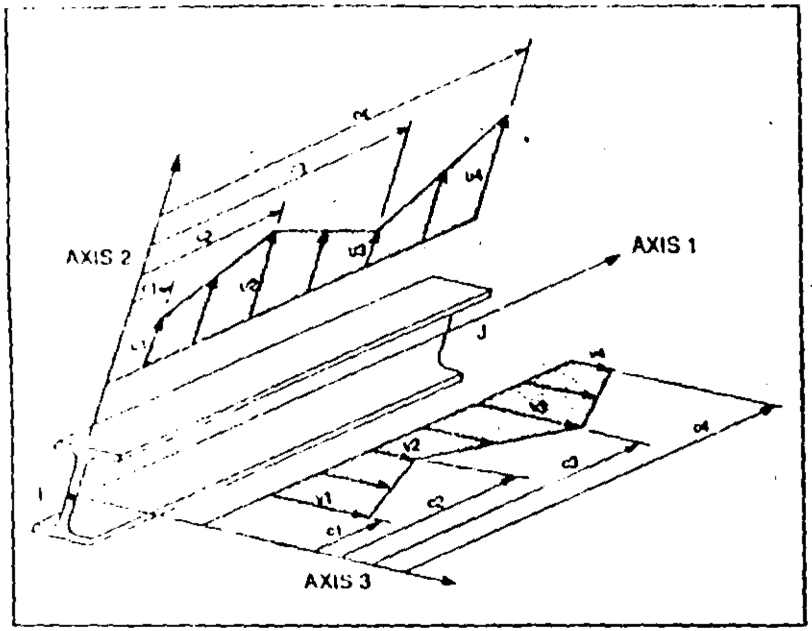


Figure X-12
Definition of Positive Trapezoidal Loads

20. Non-uniform loads are possible. There can be up to four different sections at which load intensities in each of the local 2- and 3-directions can be specified. If specified, $c_1 < c_2 \le c_3 < c_4$ Negative values of c_1, c_2, c_3 and c_4 are interpreted a ...actions of the length of the member. Negative and positive values of c_1, c_2, c_3 and c_4 should not be mixed in the same specification. See Figures X-12 and X-13.

21. The element identification number can be any number between 1 and **nid** (SYSTEM data block). Element numbers do not have to be consecutive and may be supplied in any order.

Elements may be re-specified or re-generated, in which case only the last definition is used. When an element is rede-

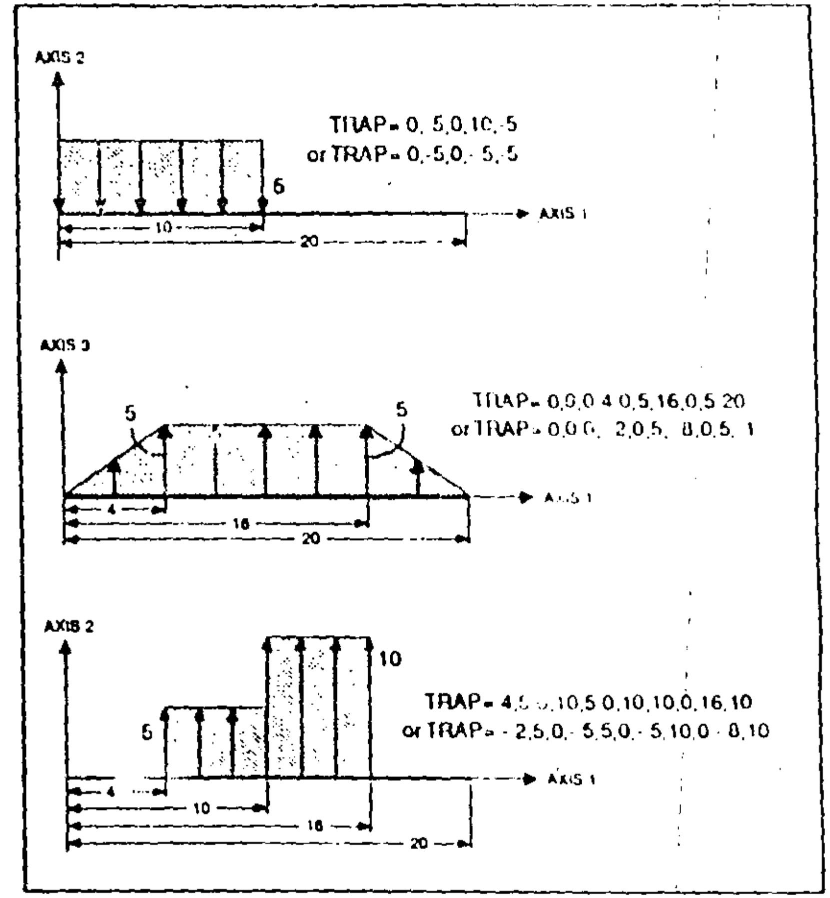


Figure X-13
Trapezoidal Load Examples

fined the previous definition is completely lost; all unspecified variables use the standard default values, and "previous-value" defaults refer to the previous data line, not to the previous definition of the element being redefined.

A previously defined element can be deleted by setting **jid** to the *negative* of its identification number. This may be used, for example, to create gaps within regions of generated elements. The only other data permitted on the data line when deleting elements is $G=n_g$, nine which specifies additional elements to be deleted. See Note 30.

22. j_i and j_j must be valid joint numbers already defined in the JOINTS data block. j_i must not have the same coordinates as j_j.

23. msp_i and msp_j refer back to the property data table defined in Format Section c. msp_i and msp_j must be positive numbers not greater than **npro**.

For prismatic members only msp_i need be specified. For non prismatic members msp_i and msp_j, the section properties at the ends I and J, respectively, must be specified. The parameter **ivar** controls the variation of $e \cdot i_{33}$ along the member length. This variation can be linear, parabolic or cubic. All other properties are assumed to vary linearly between the two ends of the member.

24. The user must clearly understand the definition of the element local 1-2-3 coordinate system in reference to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems.

Axis 1, or the V_1 vector, is always defined by the line along the axis of the member, the positive direction being directed from End I to End J. Axis 2 and Axis 3 are defined with the $LP=n_1,n_2$ option by any one of the three methods defined below.

If LP is not specified it defaults to LP=1,0 unless the element is parallel to the Z-axis, in which case it defaults to LP=2,0.

It is apparent that each element has a different local coordinate system. Element properties, loading and cross-sec-

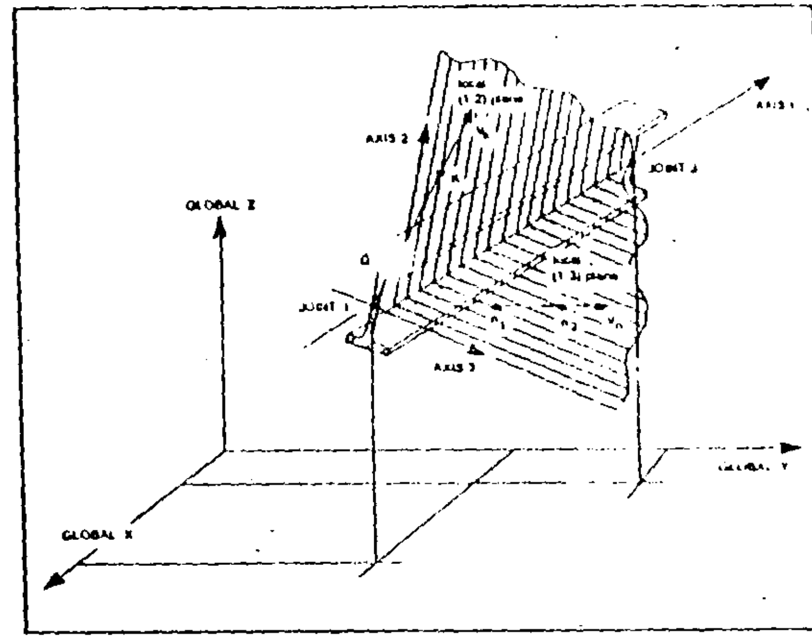


Figure X-14
Local Coordinate System for Three-dimensional Frame Members

tional forces are given in reference to this system for each element. It is up to the user to define systems which simplify data input and interpretation of results.

Method 1. Two Joint Specification, $LP=n_1,n_2$

This option is activated if both n_1 and n_2 are specified, in which case n_1 and n_2 are two joint numbers defining vector V_n as shown in Figure X-14. The vector V_n must lie in, or be parallel to, the local 1-3 plane. Axis 2 and Axis 3 are then defined by the following cross products:

$$V_2 = V_n \times V_1$$
$$V_3 = V_1 \times V_2$$

[illegible]	[illegible]	[illegible]	2L3X4X5/16-6
[illegible]	[illegible]	[illegible]	2L3X3.5X3/8
[illegible]	[illegible]	[illegible]	2L3X3.5X5/16-3
[illegible]	[illegible]	[illegible]	2L3X3.5X1/4-6
[illegible]	2L2.5X3.5X3/8-3	2L2.5X3.5X3/8-6	2L2.5X3.5X5/16
[illegible]	2L2.5X3.5X5/16-6	2L2.5X3.5X1/4	2L2.5X3.5X1/4-3
[illegible]	2L2.5X3X3/8	2L2.5X3X3/8-3	2L2.5X3X3/8-6
[illegible]	2L2.5X3X1/4-3	2L2.5X3X1/4-6	2L2.5X3X3/16
[illegible]	2L2.5X3X3/16-6	2L2X3X3/8	2L2X3X3/8-3
[illegible]	[illegible]	2L2X3X5/16-3	2L2X3X5/16-6
[illegible]	[illegible]	2L2X3X1/4-6	2L2X3X3/16
[illegible]	[illegible]	2L2X2.5X3/8	2L2X2.5X3/8-3
[illegible]	2L2X2.5X5/16	2L2X2.5X5/16-3	2L2X2.5X5/16-6
[illegible]	2L2X2.5X1/4-3	2L2X2.5X1/4-6	2L2X2.5X3/16
[illegible]	2L2X2.5X3/16-6	TS16X16X1/2	[illegible]
[illegible]	[illegible]	TS14X14X3/8	TS14X14X5/16
[illegible]	TS12X12X3/8	TS12X12X5/16	TS12X12X1/4
[illegible]	TS10X10X1/2	TS10X10X3/8	TS10X10X5/16
[illegible]	TS8X8X5/8	TS8X8X1/2	TS8X8X3/8
[illegible]	TS8X8X1/4	TS8X8X3/16	TS7X7X1/2
[illegible]	TS7X7X5/16	TS7X7X1/4	TS7X7X3/16
[illegible]	TS6X6X3/8	TS6X6X5/16	TS6X6X1/4
[illegible]	TS5X5X1/2	TS5X5X3/8	TS5X5X5/16
[illegible]	TS5X5X3/16	TS4X4X1/2	TS4X4X3/8
[illegible]	TS4X4X1/4	TS4X4X3/16	TS3.5X3.5X5/16
[illegible]	TS3.5X3.5X3/16	TS3X3X5/16	TS3X3X1/4
[illegible]	TS2.5X2.5X1/4	TS2.5X2.5X5/16	TS2X2X1/4
[illegible]	TS20X12X1/2	TS20X12X3/8	TS20X12X5/16
[illegible]	TS20X8X3/8	TS20X8X5/16	TS20X4X1/2
[illegible]	TS20X4X5/16	TS18X6X1/2	TS18X6X3/8
[illegible]	TS16X12X1/2	TS16X12X3/8	TS16X12X5/16
[illegible]	TS16X8X3/8	TS16X8X5/16	TS16X4X1/2
[illegible]	TS16X4X5/16	TS14X10X1/2	TS14X10X3/8
[illegible]	TS14X6X1/2	TS14X6X3/8	TS14X6X5/16
[illegible]	TS14X4X1/2	TS14X4X3/8	TS14X4X5/16
[illegible]	TS12X8X5/8	TS12X8X1/2	TS12X8X3/8
[illegible]	TS12X8X1/4	TS12X6X5/8	TS12X6X1/2
[illegible]	TS12X6X5/16	TS12X6X1/4	TS12X6X3/16
[illegible]	TS12X4X3/8	TS12X4X5/16	TS12X4X1/4
[illegible]	TS12X2X1/4	TS12X2X3/16	TS10X6X5/8
[illegible]	TS10X6X3/8	TS10X6X5/16	TS10X6X1/4
[illegible]	TS10X4X1/2	TS10X4X3/8	TS10X4X5/16
[illegible]	TS10X4X3/16	TS10X2X3/8	TS10X2X5/16
[illegible]	TS10X2X3/16	TS8X6X1/2	TS8X6X3/8
[illegible]	TS8X6X1/4	TS8X6X3/16	TS8X4X1/2
[illegible]	TS8X4X5/16	TS8X4X1/4	TS8X4X3/16
[illegible]	TS8X3X5/16	TS8X3X1/4	TS8X3X3/16
[illegible]	TS8X2X5/16	TS8X2X1/4	TS8X2X3/16
[illegible]	TS7X5X3/8	TS7X5X5/16	TS7X5X1/4
[illegible]	TS7X4X3/8	TS7X4X5/16	TS7X4X1/4
[illegible]	TS7X3X3/8	TS7X3X5/16	TS7X3X1/4
[illegible]	TS6X4X1/2	TS6X4X3/8	TS6X4X5/16
[illegible]	TS6X4X3/16	TS6X3X3/8	TS6X3X5/16
[illegible]	TS6X3X3/16	TS6X2X3/8	TS6X2X5/16
[illegible]	TS6X2X3/16	TS5X4X3/8	TS5X4X5/16
[illegible]	TS5X4X3/16	TS5X3X1/2	TS5X3X3/8
[illegible]	TS5X3X1/4	TS5X3X3/16	TS5X2X5/16
[illegible]	TS5X2X3/16	TS4X3X5/16	TS4X3X1/4
[illegible]	TS4X2X5/16	TS4X2X1/4	TS4X2X3/16
[illegible]	TS3X2X3/16	PS.5	PS.75
PS1	PS1.25	PS1.5	PS2
PS2.5	PS3	PS3.5	PS4
PS5	PS6	PS8	PS10
PS12	PE.5	PE.75	PE1
PE1.25	PE1.5	PE2	PE2.5
PE3	PE3.5	PE4	PE5
PE6	PE8	PE25	PE12
PE12	PD2.5	PD3	PD4
PD5	PD6	PD8	

Figure X-10 (continued)
Built-In AISC Property Designations

added to identify structural tubes, standard pipes, extra strong pipes and double extra strong pipes, respectively. Also, for the double angles the legs with the first dimension are back-to-back, and the hyphenated number at the end of the name denotes the back-to-back distance in eighths of an inch.

The convention for the major and minor axes for the AISC sections is as shown on Figure X-9 for the corresponding shapes. For the structural tubes, angles and double angles the first dimension is parallel to the 2-axis.

For this option the AISC.DAT file supplied with the program should be located in either the directory that contains the data files for this problem, or the directory that contains the SAP90 executables. If present in both directories, it will use the file in the data directory.

The AISC.DAT file is available in inch or meter units, and if this option is used consistent length units should be used throughout the input data file. See the Installation Guide [9].

If a user-defined section-property database is to be used, the user must first create the database using the program PROPER available upon request from CSI. This allows the user to conveniently use custom-built and other non-AISC sections. The user database to be used is specified as:

NAME=filename

on the Frame Control line (see Format Section b above). The **filename** is the name of the file created by PROPER with a .DAT extension. This file should be located in one of the directories described above for the AISC.DAT file.

If any explicit value for a, j, i33, i22, a2 or a3 is provided with any automatically-calculated or recovered section-property specification, it overwrites the calculated or recovered property value.

15. Span loading identification numbers must be in ascending, consecutive, numerical sequence starting with one (1).

16. w_1, w_2 and w_3 are uniform loads in force per unit of total member length (not projected length) along the positive local 1-, 2-, and 3-directions respectively. See Figure X-11. The local axes are defined in Note 24.

17. w_X, w_Y and w_Z are the uniform loads in force per unit of *projected member length* acting in the direction of the global X, Y and Z axes respectively. See Figure X-11.

18. The units of temperature should correspond to the units used in the specification of the coefficient of thermal expansion. t_1 is in degrees. t_2 and t_3 are in degrees per unit length. Thermal gradients are positive if the temperature increases (linearly) in the positive local axis direction. Gradients produce bending strains only. Axial thermal strains are computed from the temperature rise t_1. This thermal load is in addition to any thermal loads defined using the POTENTIAL data block (Note 4).

19. There can be up to four point loads specified in each of the local 2- and local 3-directions of the member. If specified, $d_1 < d_2 < d_3 < d_4$. Negative values of d_1, d_2, d_3 and d_4 are interpreted as fractions of the length of the member. Negative and positive values of d_1, d_2, d_3 and d_4 should not be mixed in the same specification. See Figure X-11.

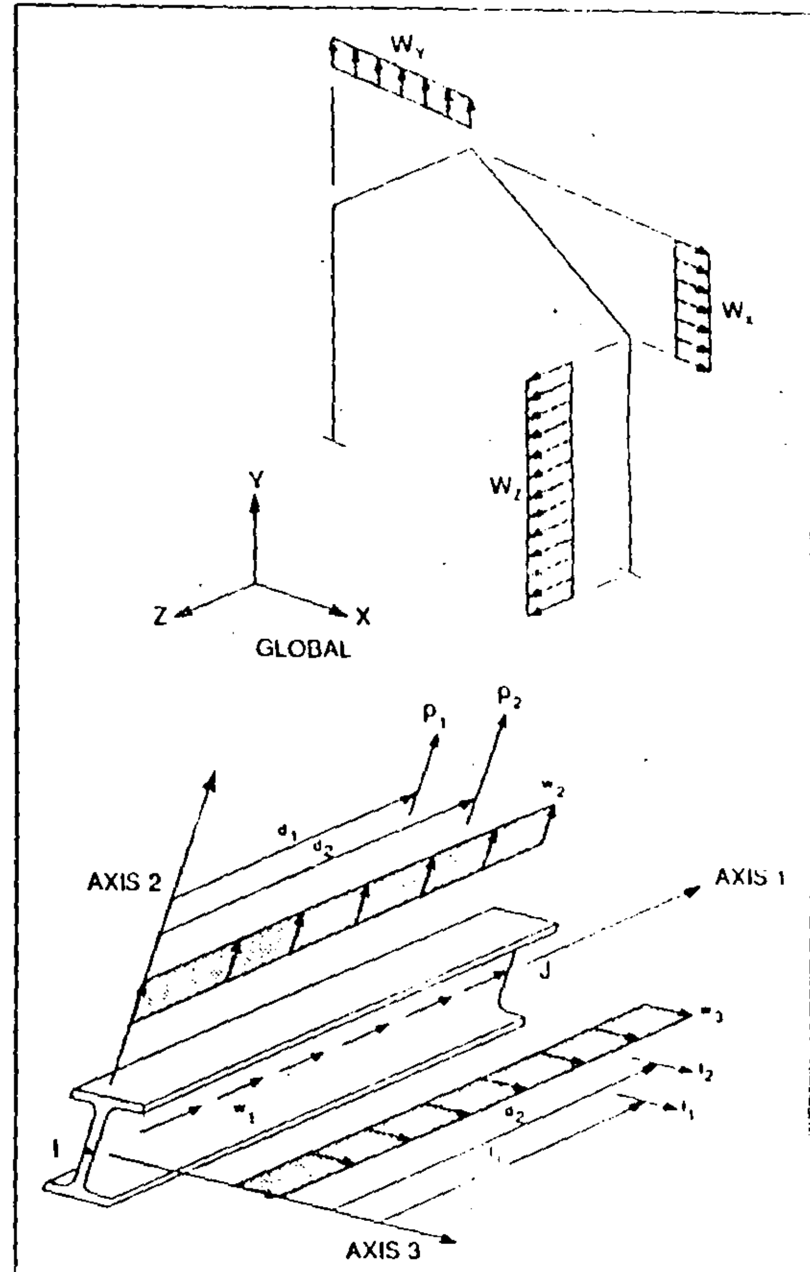


Figure X-11
Definition of Positive Point Loads and Uniform Loads

W40x328	W40x298	W40x268	W40x244
W40x221	W40x192	W40x655	W40x593
W40x531	W40x480	W40x436	W40x397
W40x362	W40x324	W40x297	W40x277
W40x249	W40x215	W40x199	W40x183
W40x167	W40x149	W36x848	W36x798
W36x720	W36x650	W36x588	W36x527
W36x485	W36x439	W36x393	W36x359
W36x328	W36x300	W36x280	W36x260
W36x245	W36x230	W36x256	W36x232
W36x210	W36x194	W36x182	W36x170
W36x160	W36x150	W36x135	W33x619
W33x567	W33x515	W33x468	W33x424
W33x387	W33x354	W33x318	W33x291
W33x263	W33x241	W33x221	W33x201
W33x169	W33x152	W33x141	W33x130
W33x118	W30x581	W30x526	W30x477
W30x433	W30x391	W30x357	W30x326
W30x292	W30x261	W30x235	W30x211
W30x191	W30x173	W30x148	W30x132
W30x124	W30x116	W30x108	W30x99
W30x90	W27x539	W27x494	W27x448
W27x407	W27x368	W27x336	W27x307
W27x281	W27x258	W27x235	W27x217
W27x194	W27x178	W27x161	W27x146
W27x129	W27x114	W27x102	W27x94
W27x84	W24x492	W24x450	W24x408
W24x370	W24x335	W24x306	W24x279
W24x250	W24x229	W24x207	W24x192
W24x176	W24x162	W24x146	W24x131
W24x117	W24x104	W24x103	W24x94
W24x84	W24x76	W24x68	W24x62
W24x55	W21x402	W21x364	W21x333
W21x300	W21x275	W21x248	W21x223
W21x201	W21x182	W21x166	W21x147
W21x132	W21x122	W21x111	W21x101
W21x93	W21x83	W21x73	W21x68
W21x62	W21x57	W21x50	W21x44
W18x311	W18x283	W18x258	W18x234
W18x211	W18x192	W18x175	W18x158
W18x143	W18x130	W18x119	W18x106
W18x97	W18x86	W18x76	W18x71
W18x65	W18x60	W18x55	W18x50
W18x46	W18x40	W18x35	W16x100
W16x89	W16x77	W16x67	W16x57
W16x50	W16x45	W16x40	W16x36
W16x31	W16x26	W14x730	W14x665
W14x605	W14x550	W14x500	W14x455
W14x426	W14x398	W14x370	W14x342
W14x311	W14x283	W14x257	W14x233
W14x211	W14x193	W14x176	W14x159
W14x145	W14x132	W14x120	W14x109
W14x99	W14x90	W14x82	W14x74
W14x68	W14x61	W14x53	W14x48
W14x43	W14x38	W14x34	W14x30
W14x26	W14x22	W12x336	W12x305
W12x279	W12x252	W12x230	W12x210
W12x190	W12x170	W12x152	W12x136
W12x120	W12x106	W12x96	W12x87
W12x79	W12x72	W12x65	W12x58
W12x53	W12x50	W12x45	W12x40
W12x35	W12x30	W12x26	W12x22
W12x19	W12x16	W12x14	W10x112
W10x100	W10x88	W10x77	W10x68
W10x60	W10x54	W10x49	W10x45
W10x39	W10x33	W10x30	W10x26
W10x22	W10x19	W10x17	W10x15
W10x12	W8x67	W8x58	W8x48
W8x40	W8x35	W8x31	W8x28
W8x24	W8x21	W8x18	W8x15
W8x13	W8x10	W6x25	W6x20

Figure X-10

Built-In AISC Property Designations

W6x15	W6x16	W6x12	W6x9
W5x19	W5x16	W4x13	M14x18
M12x11.8	M10x9	M8x6.5	M6x20
M6x4.4	M5x18.9	M4x13	S24x121
S24x106	S24x100	S24x90	S24x80
S20x96	S20x86	S20x75	S20x66
S18x70	S18x54.7	S15x50	S15x42.9
S12x50	S12x40.8	S12x35	S12x31.8
S10x35	S10x25.4	S8x23	S8x18.4
S7x20	S7x15.3	S6x17.25	S6x12.5
S5x14.75	S5x10	S4x9.5	S4x7.7
S3x7.5	S3x5.7	HP14x117	HP14x102
HP14x89	HP14x73	HP13x100	HP13x87
HP13x73	HP13x60	HP12x84	HP12x74
HP12x63	HP12x53	HP10x57	HP10x42
HP8x36	C15x50	C15x40	C15x33.9
C12x30	C12x25	C12x20.7	C10x30
C10x25	C10x20	C10x15.3	C9x20
C9x15	C9x13.4	C8x18.75	C8x13.75
C8x11.5	C7x14.75	C7x12.25	C7x9.8
C6x13	C6x10.5	C6x8.2	C5x9
C5x6.7	C4x7.25	C4x5.4	C3x6
C3x5	C3x4.1	MC18x58	MC18x51.9
MC18x45.8	MC18x42.7	MC13x50	MC13x40
MC13x35	MC13x31.8	MC12x50	MC12x45
MC12x40	MC12x35	MC12x31	MC12x10.6
MC10x41.1	MC10x33.6	MC10x28.5	MC10x25
MC10x22	MC10x8.4	MC10x6.5	MC9x25.4
MC9x23.9	MC8x22.8	MC8x21.4	MC8x20
MC8x18.7	MC8x8.5	MC7x22.7	MC7x19.1
MC7x17.6	MC6x18	MC6x15.3	MC6x16.3
MC6x15.1	MC6x12	WT18x179.5	WT18x164
WT18x150	WT18x140	WT18x130	WT18x122.5
WT18x115	WT18x128	WT18x116	WT18x105
WT18x97	WT18x91	WT18x85	WT18x80
WT18x75	WT18x67.5	WT16.5x177	WT16.5x159
WT16.5x145.5	WT16.5x131.5	WT16.5x120.5	WT16.5x110.5
WT16.5x100.5	WT16.5x84.5	WT16.5x76	WT16.5x70.5
WT16.5x65	WT16.5x59	WT15x117.5	WT15x105.5
WT15x95.5	WT15x86.5	WT15x76	WT15x66
WT15x62	WT15x58	WT15x54	WT15x49.5
WT13.5x108.5	WT13.5x97	WT13.5x89	WT13.5x80.5
WT13.5x73	WT13.5x64.5	WT13.5x57	WT13.5x51
WT13.5x47	WT13.5x42	WT12x88	WT12x81
WT12x73	WT12x65.5	WT12x58.5	WT12x52
WT12x51.5	WT12x47	WT12x42	WT12x38
WT12x34	WT12x31	WT12x27.5	WT10.5x83
WT10.5x73.5	WT10.5x66	WT10.5x61	WT10.5x55.5
WT10.5x50.5	WT10.5x46.5	WT10.5x41.5	WT10.5x36.5
WT10.5x34	WT10.5x31	WT10.5x28.5	WT10.5x25
WT10.5x22	WT9x71.5	WT9x65	WT9x59.5
WT9x53	WT9x48.5	WT9x43	WT9x38
WT9x35.5	WT9x32.5	WT9x30	WT9x27.5
WT9x25	WT9x23	WT9x20	WT9x17.5
WT8x50	WT8x44.5	WT8x38.5	WT8x33.5
WT8x28.5	WT8x25	WT8x22.5	WT8x20
WT8x18	WT8x15.5	WT8x13	WT7x365
WT7x332.5	WT7x302.5	WT7x275	WT7x250
WT7x227.5	WT7x213	WT7x199	WT7x185
WT7x171	WT7x155.5	WT7x141.5	WT7x128.5
WT7x116.5	WT7x105.5	WT7x96.5	WT7x88
WT7x79.5	WT7x72.5	WT7x66	WT7x60
WT7x54.5	WT7x49.5	WT7x45	WT7x41
WT7x37	WT7x34	WT7x30.5	WT7x26.5
WT7x24	WT7x21.5	WT7x19	WT7x17
WT7x15	WT7x13	WT7x11	WT6x168
WT6x152.5	WT6x139.5	WT6x126	WT6x115
WT6x105	WT6x95	WT6x85	WT6x76
WT6x68	WT6x60	WT6x53	WT6x48
WT6x43.5	WT6x39.5	WT6x36	WT6x32.5
WT6x29	WT6x26.5	WT6x25	WT6x22.5

Figure X-10 (continued)

Built-In AISC Property Designations

[illegible]	[illegible]	MT6x15	MT6x13
[illegible]	[illegible]	[illegible]	MT6x7
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	MT4x15.5
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	MT2.5x9.5	MT2.5x8	MT2x6.5
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	[illegible]	[illegible]	[illegible]
[illegible]	ST12x53	ST12x50	ST12x45
[illegible]	ST10x48	ST10x43	ST10x37.5
[illegible]	ST9x35	[illegible]	ST7.5x25
[illegible]	ST6x25	ST6x20.4	ST6x17.5
[illegible]	ST5x17.5	ST5x12.7	ST4x11.5
[illegible]	ST3.5x10	ST3.5x7.65	ST3x8.625
[illegible]	ST2.5x7.375	ST2.5x5	ST2x6.75
[illegible]	ST1.5x3.75	ST1.5x2.85	L9x4x5/8
L9x4x9/16	[illegible]	L8x8x9/8	L8x8x1
[illegible]	L8x8x3/4	L8x8x5/8	L8x8x9/16
[illegible]	[illegible]	L8x6x7/8	L8x6x3/4
[illegible]	L8x6x9/16	L8x6x1/2	L8x6x7/16
[illegible]	L8x4x3/4	L8x4x7/16	L8x4x1/2
[illegible]	L7x4x5/8	L7x4x1/2	L7x4x3/8
[illegible]	L6x6x7/8	L6x6x3/4	L6x6x5/8
[illegible]	L6x6x1/2	L6x6x7/16	L6x6x3/8
[illegible]	L6x4x7/8	L6x4x3/4	L6x4x5/8
L6x4x9/16	L6x4x1/2	L6x4x7/16	L6x4x3/8
[illegible]	L6x3.5x1/2	L6x3.5x3/8	L6x3.5x5/16
[illegible]	L5x5x3/4	L5x5x5/8	L5x5x1/2
[illegible]	L5x5x3/8	L5x5x5/16	L5x3.5x3/4
[illegible]	L5x3.5x1/2	L5x3.5x7/16	L5x3.5x3/8
[illegible]	L5x3.5x1/4	L5x3x5/8	L5x3x1/2
[illegible]	L5x3x3/8	L5x3x5/16	L5x3x1/4
[illegible]	L4x4x5/8	L4x4x1/2	L4x4x7/16
[illegible]	L4x4x5/16	L4x4x1/4	L4x3.5x5/8
[illegible]	L4x3.5x7/16	L4x3.5x3/8	L4x3.5x5/16
[illegible]	L4x3x5/8	L4x3x1/2	L4x3x7/16
[illegible]	L4x3x5/16	L4x3x1/4	L3.5x3.5x1/2
[illegible]	L3.5x3.5x3/8	L3.5x3.5x5/16	L3.5x3.5x1/4
[illegible]	L3.5x3x7/16	L3.5x3x3/8	L3.5x3x5/16
[illegible]	L3.5x2.5x1/2	L3.5x2.5x7/16	L3.5x2.5x3/8
[illegible]	L3.5x2.5x1/4	L3x3x1/2	L3x3x7/16
L3x3x3/8	L3x3x5/16	L3x3x1/4	L3x3x3/16
L3x2.5x1/2	L3x2.5x7/16	L3x2.5x3/8	L3x2.5x5/16
L3x2.5x1/4	L3x2.5x3/16	L3x2x1/2	L3x2x7/16
L3x2x3/8	L3x2x5/16	L3x2x1/4	L3x2x3/16
L2.5x2.5x1/2	L2.5x2.5x3/8	L2.5x2.5x5/16	L2.5x2.5x1/4
L2.5x2.5x3/16	L2.5x2x3/8	L2.5x2x5/16	L2.5x2x1/4
L2.5x2x3/16	L2x2x3/8	L2x2x5/16	L2x2x1/4
L2x2x3/16	L2x2x1/8	[illegible]	2L8x8x9/8-3
2L8x8x9/8-6	2L8x8x1	[illegible]	2L8x8x1-6
2L8x8x7/8	2L8x8x7/8-3	2L8x8x7/8-6	2L8x8x3/4
2L8x8x3/4-3	2L8x8x3/4-6	2L8x8x5/8	2L8x8x5/8-3
2L8x8x5/8-6	2L8x8x1/2	2L8x8x1/2-3	2L8x8x1/2-6
2L6x6x1	2L6x6x1-3	2L6x6x1-6	2L6x6x7/8
2L6x6x7/8-3	2L6x6x7/8-6	2L6x6x3/4	2L6x6x3/4-3
2L6x6x3/4-6	2L6x6x5/8	2L6x6x5/8-3	2L6x6x5/8-6
2L6x6x1/2	2L6x6x1/2-3	2L6x6x1/2-6	2L6x6x3/8
2L6x6x3/8-3	2L6x6x3/8-6	2L5x5x7/8	2L5x5x7/8-3
2L5x5x7/8-6	2L5x5x3/4	2L5x5x3/4-3	2L5x5x3/4-6
2L5x5x1/2	2L5x5x1/2-3	2L5x5x1/2-6	2L5x5x3/8
2L5x5x3/8-3	2L5x5x3/8-6	2L5x5x5/16	2L5x5x5/16-3
2L5x5x5/16-6	2L4x4x3/4	2L4x4x3/4-3	2L4x4x3/4-6
2L4x4x5/8	2L4x4x5/8-3	2L4x4x5/8-6	2L4x4x1/2
2L4x4x1/2-3	2L4x4x1/2-6	2L4x4x3/8	2L4x4x3/8-3
2L4x4x3/8-6	[illegible]	2L4x4x5/16-3	2L4x4x5/16-6

Figure X-10 (continued)
Built-In AISC Property Designations

2L4x4x1/4	2L4x4x1/4-3	2L4x4x1/4-6	2L3.5x3.5x1/2
2L3.5x3.5x3/8-3	2L3.5x3.5x3/8-6	2L3.5x3.5x5/16	2L3.5x3.5x5/16-3
2L3.5x3.5x5/16-6	2L3.5x3.5x1/4	2L3.5x3.5x1/4-3	2L3.5x3.5x1/4-6
2L3x3x1/2	2L3x3x1/2-3	2L3x3x1/2-6	2L3x3x3/8
2L3x3x3/8-3	2L3x3x3/8-6	2L3x3x5/16	2L3x3x5/16-3
2L3x3x5/16-6	2L3x3x1/4	2L3x3x1/4-3	2L3x3x1/4-6
2L3x3x3/16	2L3x3x3/16-3	2L3x3x3/16-6	2L2.5x2.5x3/8
2L2.5x2.5x3/8-3	2L2.5x2.5x3/8-6	2L2.5x2.5x5/16	2L2.5x2.5x5/16-3
2L2.5x2.5x5/16-6	2L2.5x2.5x1/4	2L2.5x2.5x1/4-3	2L2.5x2.5x1/4-6
2L2.5x2.5x3/16	2L2.5x2.5x3/16-3	2L2.5x2.5x3/16-6	2L2x2x3/8
2L2x2x3/8-3	2L2x2x3/8-6	2L2x2x5/16	2L2x2x5/16-3
2L2x2x5/16-6	2L2x2x1/4	2L2x2x1/4-3	2L2x2x1/4-6
2L2x2x3/16	2L2x2x3/16-3	2L2x2x3/16-6	2L2x2x1/8
2L2x2x1/8-3	2L2x2x1/8-6	2L8x6x1	2L8x6x1-3
2L8x6x1-6	2L8x6x3/4	2L8x6x3/4-3	2L8x6x3/4-6
2L8x6x1/2	2L8x6x1/2-3	2L8x6x1/2-6	2L8x4x1
2L8x4x1-3	2L8x4x1-6	2L8x4x3/4	2L8x4x3/4-3
2L8x4x3/4-6	2L8x4x1/2	2L8x4x1/2-3	2L8x4x1/2-6
2L7x4x3/4	2L7x4x3/4-3	2L7x4x3/4-6	2L7x4x1/2
2L7x4x1/2-3	2L7x4x1/2-6	2L7x4x3/8	2L7x4x3/8-3
2L7x4x3/8-6	2L6x4x3/4	2L6x4x3/4-3	2L6x4x3/4-6
2L6x4x5/8	2L6x4x5/8-3	2L6x4x5/8-6	2L6x4x1/2
2L6x4x1/2-3	2L6x4x1/2-6	2L6x4x3/8	2L6x4x3/8-3
2L6x4x3/8-6	2L6x3.5x3/8	2L6x3.5x3/8-3	2L6x3.5x3/8-6
2L6x3.5x5/16	2L6x3.5x5/16-3	2L6x3.5x5/16-6	2L5x3.5x3/4
2L5x3.5x3/4-3	2L5x3.5x3/4-6	2L5x3.5x1/2	2L5x3.5x1/2-3
2L5x3.5x1/2-6	2L5x3.5x3/8	2L5x3.5x3/8-3	2L5x3.5x3/8-6
2L5x3.5x5/16	2L5x3.5x5/16-3	2L5x3.5x5/16-6	2L5x3x1/2
2L5x3x1/2-3	2L5x3x1/2-6	2L5x3x3/8	2L5x3x3/8-3
2L5x3x3/8-6	2L5x3x5/16	2L5x3x5/16-3	2L5x3x5/16-6
2L5x3x1/4	2L5x3x1/4-3	2L5x3x1/4-6	2L4x3.5x1/2
2L4x3.5x1/2-3	2L4x3.5x1/2-6	2L4x3.5x3/8	2L4x3.5x3/8-3
2L4x3.5x3/8-6	2L4x3.5x5/16	2L4x3.5x5/16-3	2L4x3.5x5/16-6
2L4x3.5x1/4	2L4x3.5x1/4-3	2L4x3.5x1/4-6	2L4x3x1/2
2L4x3x1/2-3	2L4x3x1/2-6	2L4x3x3/8	2L4x3x3/8-3
2L4x3x3/8-6	2L4x3x5/16	2L4x3x5/16-3	2L4x3x5/16-6
2L4x3x1/4	2L4x3x1/4-3	2L4x3x1/4-6	2L3.5x3x3/8
2L3.5x3x3/8-3	2L3.5x3x3/8-6	2L3.5x3x5/16	2L3.5x3x5/16-3
2L3.5x3x5/16-6	2L3.5x3x1/4	2L3.5x3x1/4-3	2L3.5x3x1/4-6
2L3.5x2.5x3/8	2L3.5x2.5x3/8-3	2L3.5x2.5x3/8-6	2L3.5x2.5x5/16
2L3.5x2.5x5/16-3	2L3.5x2.5x5/16-6	2L3.5x2.5x1/4	2L3.5x2.5x1/4-3
2L3.5x2.5x1/4-6	2L3x2.5x3/8	2L3x2.5x3/8-3	2L3x2.5x3/8-6
2L3x2.5x1/4	2L3x2.5x1/4-3	2L3x2.5x1/4-6	2L3x2.5x3/16
2L3x2.5x3/16-3	2L3x2.5x3/16-6	2L3x2x3/8	2L3x2x3/8-3
2L3x2x3/8-6	2L3x2x5/16	2L3x2x5/16-3	2L3x2x5/16-6
2L3x2x1/4	2L3x2x1/4-3	2L3x2x1/4-6	2L3x2x3/16
2L3x2x3/16-3	2L3x2x3/16-6	2L2.5x2x3/8	2L2.5x2x3/8-3
2L2.5x2x3/8-6	2L2.5x2x5/16	2L2.5x2x5/16-3	2L2.5x2x5/16-6
2L2.5x2x1/4	2L2.5x2x1/4-3	2L2.5x2x1/4-6	2L2.5x2x3/16
2L2.5x2x3/16-3	2L2.5x2x3/16-6	2L6x8x1	2L6x8x1-3
2L6x8x1-6	2L6x8x3/4	2L6x8x3/4-3	2L6x8x3/4-6
2L6x8x1/2	2L6x8x1/2-3	2L6x8x1/2-6	2L4x8x1
2L4x8x1-3	2L4x8x1-6	2L4x8x3/4	2L4x8x3/4-3
2L4x8x3/4-6	2L4x8x1/2	2L4x8x1/2-3	2L4x8x1/2-6
2L4x7x3/4	2L4x7x3/4-3	2L4x7x3/4-6	2L4x7x1/2
2L4x7x1/2-3	2L4x7x1/2-6	2L4x7x3/8	2L4x7x3/8-3
2L4x7x3/8-6	2L4x6x3/4	2L4x6x3/4-3	2L4x6x3/4-6
2L4x6x5/8	2L4x6x5/8-3	2L4x6x5/8-6	2L4x6x1/2
2L4x6x1/2-3	2L4x6x1/2-6	2L4x6x3/8	2L4x6x3/8-3
2L4x6x3/8-6	2L3.5x6x3/8	2L3.5x6x3/8-3	2L3.5x6x3/8-6
2L3.5x6x5/16	2L3.5x6x5/16-3	2L3.5x6x5/16-6	2L3.5x5x3/4
2L3.5x5x3/4-3	2L3.5x5x3/4-6	2L3.5x5x1/2	2L3.5x5x1/2-3
2L3.5x5x1/2-6	2L3.5x5x3/8	2L3.5x5x3/8-3	2L3.5x5x3/8-6
2L3.5x5x5/16	2L3.5x5x5/16-3	2L3.5x5x5/16-6	2L3x5x1/2
2L3x5x1/2-3	2L3x5x1/2-6	2L3x5x3/8	2L3x5x3/8-3
2L3x5x3/8-6	2L3x5x5/16	2L3x5x5/16-3	2L3x5x5/16-6
2L3x5x1/4	2L3x5x1/4-3	2L3x5x1/4-6	2L3.5x4x1/2
2L3.5x4x1/2-3	2L3.5x4x1/2-6	2L3.5x4x3/8	2L3.5x4x3/8-3
2L3.5x4x3/8-6	2L3.5x4x5/16	2L3.5x4x5/16-3	2L3.5x4x5/16-6
2L3.5x4x1/4	2L3.5x4x1/4-3	2L3.5x4x1/4-6	2L3x4x1/2
2L3x4x1/2-3	2L3x4x1/2-6	2L3x4x3/8	2L3x4x3/8-3

Figure X-10 (continued)
Built-In AISC Property Designations

forces along the local Axis 3. Also see Note 14 below. The local axis convention is defined in Note 24.

A shear area of pure zero will cause the program to exclude the effect of shear deformations. In other words, the shear deformations will be assumed to be zero. Effectively, a pure zero shear area is defaulted to an infinite shear area by the program. Formulae for calculating the shear areas of typical sections are given in Figure X-8.

11. This is the modulus of elasticity of the material. Remember to use consistent force and length units.

12. g is the shear modulus, used for torsional and shear components, and is related to the modulus of elasticity by

$$g = \frac{e}{2\,(1 + u)}$$

where u is Poisson's ratio.

13. The weight per unit length is used for the self weight calculation of the structure. The self weight is added into the structural load conditions via the gravitational multipliers described in Note 4.

The mass per unit length is used for the calculation of the mass of the element. Consistent mass units must be used. This entry is only necessary in a dynamic analysis mode for automatic lumping of the element mass to the element joints when assembling the structural mass matrix.

14. The basic input format for *section* properties of a FRAME element is of the following form:

A=a J=j I=i33,i22 AS=a2,a3

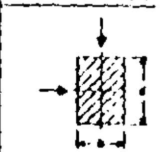
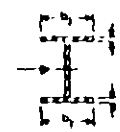
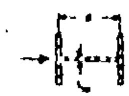
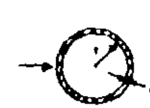
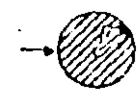
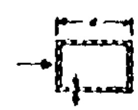
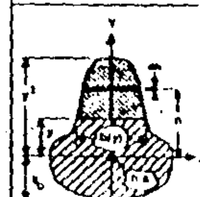
Section	Description	Effective Shear Area
	Rectangular Section Shear Forces parallel to the b or d directions	$\frac{5}{6}\,bd$
	Wide Flange Section Shear Forces parallel to flange	$\frac{5}{3}\,t_f b_f$
	Wide Flange Section Shear Forces parallel to web	$t_w d$
	Thin Walled Circular Tube Section Shear Forces from any direction	$\pi r t$
	Solid Circular Section Shear Forces from any direction	$0.9\,\pi r^2$
	Thin Walled Rectangular Tube Section Shear Forces parallel to d-direction	$2\,t\,d$
	General Section Shear Forces parallel to Y-direction I_x = moment of inertia of section about X-X $Q(y) = \int_{y}^{y_t} \eta\, b(\eta)\, d\eta$	$\dfrac{I_x^2}{\int_{y_b}^{y_t} \frac{Q^2(y)}{b(y)}\, dy}$

Figure X-8
Shear Area Formulas

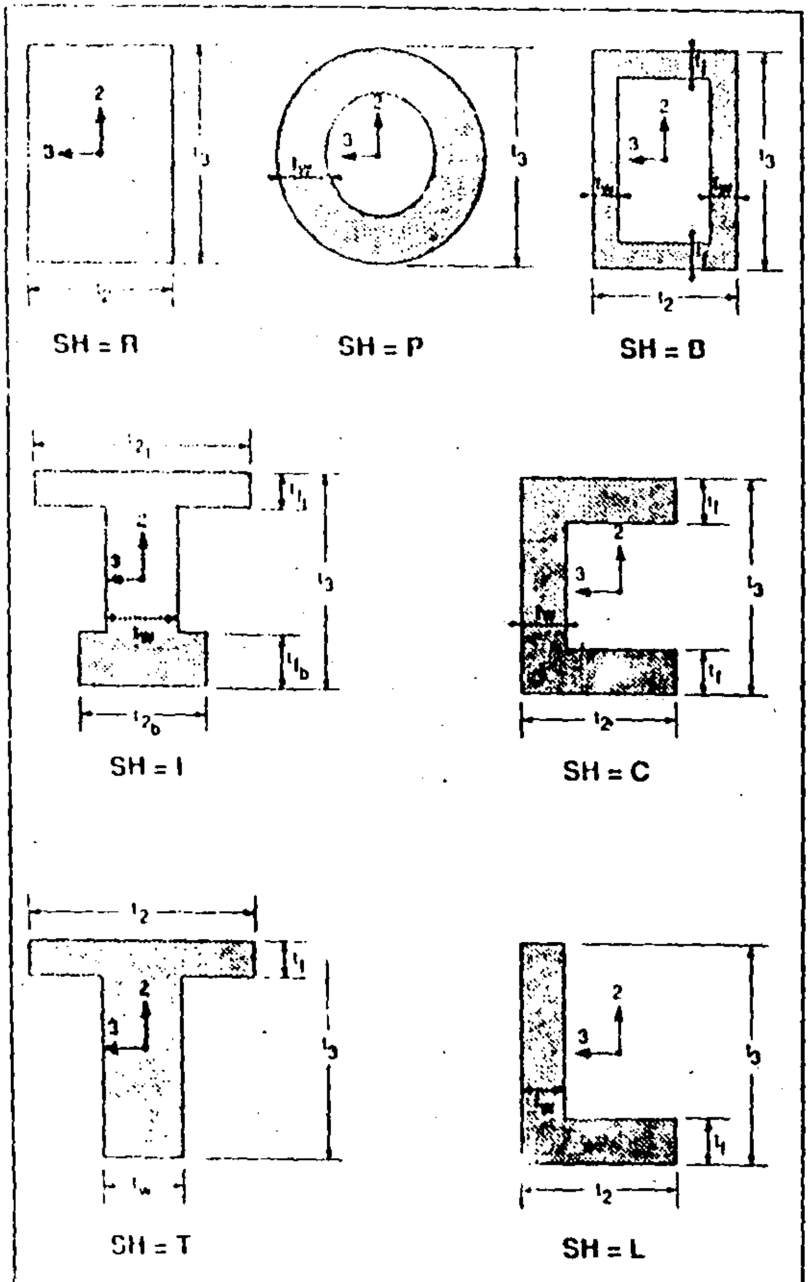


Figure X-9
Automatic Section Property Calculation

For elements having the geometric shapes shown in Figure X-9, these properties need not be evaluated. The above six properties are automatically calculated by the program if the shape and dimensional information is specified in place of the above input in one of the following forms:

SH=R T= t_3, t_2

SH=P T= t_3, t_w

SH=B T= t_3, t_2, t_f, t_w

SH=I T= t_3, t_{2t}, t_{ft}, t_w, t_{2b}, t_{fb}
(if t_{2b} and t_{fb} are 0.0, they are taken equal to t_{2t} and t_{ft}, respectively)

SH=C T = t_3, t_2, t_f, t_w

SH=T T = t_3, t_2, t_f, t_w

SH=L T = t_3, t_2, t_f, t_w

where the definitions of t_3, t_2, t_f, t_w, t_{2t}, t_{2b}, t_{ft}, t_{fb} for the various cross-sections are shown in Figure X-9.

The section properties can also be automatically recovered from the SAP90 AISC database or a user-defined database by giving the name of the desired section in the following form:

SH= AISC name (or user-defined label)
e.g., SH=W27x94

The acceptable AISC names are given in Figure X-10. The naming convention is similar to the AISC designations except that starting characters of TS, PS, PE and PD are

Variable	Note	Default	Entry
Frame Element Location Data			
nel	(21)		Element identification number
j_I	(22)		Joint number at End I
j_J			Joint number at End J
msp_I	(23)	{pv}	Property identification number at End I
msp_J		{msp_I}	Property identification number at End J
i_{var}		{2}	Variation of e • I_{33} along length of element: =1 Linear =2 Parabolic =3 Cubic
n_1,n_2	(24)	{pv}	Joint numbers to define local axis in 3-direction
r_1	(25)	{0}	End I, moment 3 release code
r_2			End J, moment 3 release code
r_3			Axial force release code
r_4			End I, moment 2 release code
r_5			End J, moment 2 release code
r_6			Torsional moment release code

Variable	Note	Default	Entry
Frame Element Location Data (continued)			
r_I,r_J	(26)	{0}	Rigid zone offsets for Ends I and J, respectively
z	(27)	{pv}	Rigid zone reduction factor
m_I,m_J	(28)	{pv}	Master joint for Ends I and J, respectively
$l_1,l_2,...$	(29)	{0}	Beam span loading patterns for load conditions l_1 through l_{nld}
n_g	(30)	{0}	Number of additional elements to be generated
ninc			Increment for element identification number
g_1			Increment for joint number at End I
g_2			Increment for joint number at End J
g_3			Increment for master joint number for End I
g_4			Increment for master joint number for End J

NOTES

1. The control parameter **npro** defines the number of data lines the program expects to read in the material and section property data section (Format Section c).

2. The control paramet... **nlost** defines the number of data lines the program expects to read in the span loading data section (Format Section d).

3. The **nsec** parameter specifies the number of equally spaced sections along the clear length of the element where the moments and shears are output. If not specified, the program automatically outputs moments and shears at the end of the clear length of the element, at maximum moment points and under point loads. **nsec** should be less than 50. Axial force and torsional moments are always output at the two ends of the element. See Notes 26 and 27.

4. The **nld** load multipliers associated with the X-direction ($x_1, x_2, \ldots, x_{nld}$) correspond to the **nld** structural load conditions. These are gravitational multipliers that activate the self weight of the FRAME elements in the X-direction. In other words, static loads acting in the X-direction, equal to the self weight of the FRAME elements, factored by the gravitational multipliers, will be added to the corresponding load condition.

   For example, if $x_2 = 1.4$, static loads consisting of 1.4 times the weight of the FRAME elements, acting in the positive global X-direction, will be added to structural load condition 2. Note that only the elements that have nonzero weights per unit length contribute to the static vectors. See Note 13. Similarly, the **nld** Y-direction multipliers generate self load vectors in the Y-direction, and the **nld** Z-direction multipliers generate self loads in the Z-direction.

   The **nld** temperature multipliers ($t_1, t_2, \ldots, t_{nld}$) similarly generate thermal body forces for the corresponding load conditions using the joint temperature values specified in the POTENTIAL data block. The zero stress, reference temperature for each element is assumed to be zero. This thermal load is in addition to any thermal loads defined as part of the span loading data (Note 18).

5. The **nld** multipliers $pr_1, pr_2, \ldots, pr_{nld}$ are prestress multipliers corresponding to the **nld** structural load conditions. Prestressing loads are applied to the frame elements via the PRESTRESS data block. The prestress multipliers define which of the structural load conditions are to receive the prestress loads. Thus if $pr_3 = 1.0$, the prestress loads (times 1.0) will be added into the load vector associated with load condition 3.

6. Property identification numbers must be in ascending, consecutive, numerical sequence starting with one (1).

7. This is the cross-sectional area of the section. Also see Note 14 below.

8. This is the torsional constant associated with the section. Also see Note 14 below.

9. i_{33} is the moment of inertia of the section about the local Axis 3, i_{22} is the moment of inertia of the section about the local Axis 2. Also see Note 14 below. The local axis convention is defined in Note 24.

10. a_2 is the shear area associated with shear forces along the local Axis 2, and a_3 is the shear area associated with shear

c. **Material and Section Property Data**

Provide **npro** data lines in this data section to define the **npro** property types in the following form:

np A=a J=j I=i_{33}, i_{22} AS=a_2, a_3
E=e G=g W=w M=m TC=alpha

d. **Span Loading Data**

Provide **nbsl** data lines in this data section, one for each of the **nbsl** span loading patterns, in the following form:

ns WL=w_1, w_2, w_3 WG=w_x, w_y, w_z T=t_1, t_2, t_3
PLD=d_1, p_1, f_1, d_2, p_2, f_2,..., d_4, p_4, f_4
TRAP=c_1, u_1, v_1, c_2, u_2, v_2, c_3, u_3, v_3, c_4, u_4, v_4

e. **FRAME Element Location Data**

In this data section provide as many data lines as needed to define all of the FRAME elements in the model. **End this data section with a blank line.** Prepare the data in the following form:

nel j_i j_j M=msp_i, msp_j, i_{var} LP=n_1, n_2
LR=r_1, r_2, r_3, r_4, r_5, r_6 RE=r_i, r_j RZ=z
MS=m_i, m_j NSL=l_1, l_2,..., l_{nld}
G=n_g, ninc, g_1, g_2, g_3, g_4

DESCRIPTION

Variable	Note	Default	Entry
Frame Control Information			
npro	(1)	[1]	Number of property types
nbsl	(2)	[0]	Number of span loading patterns
nsec	(3)	[0]	Number of force output sections
x_1,x_2,...	(4)	[0]	X-direction gravitational multipliers
y_1,y_2,...		[0]	Y-direction gravitational multipliers
z_1,z_2,...		[0]	Z-direction gravitational multipliers
t_1,t_2,...		[0]	Temperature multipliers
pr_1,pr_2,...	(5)	[0]	Prestress loading multipliers

Variable	Note	Default	Entry
Material and Section Property Data			
np	(6)		Property identification number
a	(7,14)	[0]	Area
j	(8,14)	[0]	Torsional constant
i_{33}, i_{22}	(9,14)	[0]	Moments of inertia
a_2, a_3	(10,14)	[0]	Shear areas
e	(11)	[pv]	Modulus of elasticity
g	(12)	[e/2.6]	Shear modulus
w	(13)	[0]	Weight per unit length
m	(13)	[0]	Mass per unit length
alpha	(18)	[0]	Coefficient of thermal expansion of the material (L/L/H units)
Span Loading Data			
ns	(15)		Span loading identification number
w_1	(16)	[0]	Uniform load in 1-direction - force/length
w_2			Uniform load in 2-direction - force/length
w_3			Uniform load in 3-direction - force/length

Variable	Note	Default	Entry
Span Loading Data (continued)			
w_x	(17)	[0]	Uniform load in X-direction - force/length of X-projection
w_y			Uniform load in Y-direction - force/length of Y-projection
w_z			Uniform load in Z-direction - force/length of Z-projection
t_1	(18)	[0]	Temperature increase at centerline of member
t_2			Temperature gradient in 2-direction
t_3			Temperature gradient in 3-direction
d_1	(19)		Distance from End I of p_1 and f_1
p_1			p_1 is first load from I in 2-direction
f_1			f_1 is first load from I in 3-direction
c_1	(20)		Distance from End I of u_1 and v_1
u_1			u_1 is the load intensity in the 2-direction at distance c_1 from End I
v_1			v_1 is the load intensity in the 3-direction at distance c_1 from End I

DESCRIPTION

Variable	Note	Default	Entry
Dependent joint numbers			
j_1	(1)		First dependent joint number
j_2		[j_1]	Last dependent joint number
inc		[1]	Dependent joint number increment
	Independent joint numbers for:		
c_{ux}	(2)		X-translation of dependent joint
c_{uy}			Y-translation of dependent joint
c_{uz}			Z-translation of dependent joint
c_{rx}			X-rotation of dependent joint
c_{ry}			Y-rotation of dependent joint
c_{rz}			Z-rotation of dependent joint
	Independent joint number increments for:		
i_{ux}	(2)	[0]	Joint c_{ux}
i_{uy}			Joint c_{uy}
i_{uz}			Joint c_{uz}
i_{rx}			Joint c_{rx}
i_{ry}			Joint c_{ry}
i_{rz}			Joint c_{rz}

NOTES

1. The parameters **j_1, j_2** and **inc** define the following series of joint numbers:

   j_1, j_1+inc, j_1+2inc, j_1+3inc,...

   which continues until **j_2** is reached.

2. **c_{ux}** is a joint number. The X-translation of joint **j_1** will be set equal to the X-translation of joint **c_{ux}**.

   i_{ux} is a joint number increment. If **i_{ux}** = 0, the X-translation of all the joints defined by the above series (**j_1, j_2, inc**) will be set to the X-translation of joint **c_{ux}**.

   If **i_{ux}** is not zero, the X-translation of joint **j_1** will be set equal to the X-translation of joint **c_{ux}**, and the X-translation of joint **j_1+inc** will be set equal to the X-translation of joint **c_{ux}+i_{ux}**, and so on. If **c_{ux}** is zero then no constraint is activated along the UX degree of freedom of the dependent joint.

   Similarly, **c_{uy}** is a joint number and the Y-translation of joint **j_1** will be set equal to the Y-translation of joint **c_{uy}**, and the Y-translation of joint **j_1+inc** will be set equal to the Y-translation of joint **c_{uy}+i_{uy}**, and so on.

   Similarly, **c_{uz}** corresponds to the Z-translation and **c_{rx}**, **c_{ry}** and **c_{rz}** correspond to the rotations about the X, Y and Z axes respectively.

   For example, the specification

   21 31 1 C=20,0,0,0,0,0

will cause the X-translations of joints 21 through 31 to be set equal (constrained) to the X-translations of joint number 20. The option is very useful if axial deformations are to be neglected in horizontal or vertical members.

Also, for example, the specification

40 50 1 C=0,0,10,0,0,0 I=0,0,1,0,0

will cause the Z-translations of joints 40 through 50 to be set equal to the Z-translations of joints 10 through 20, respectively. This can be used to create a shear free boundary in a finite element mesh.

A degree of freedom may only be constrained to an independent (or unconstrained) degree of freedom.

Also, an independent degree of freedom may only be constrained if it has no dependent degrees of freedom.

Repeated joint specifications are allowed; however, a degree of freedom that has been constrained by a previous specification of the joint cannot be changed by a subsequent re-specification.

A degree of freedom that has not been constrained by a previous specification of the joint can be constrained by subsequent constraint specifications.

Joint loads and masses that are applied along dependent degrees of freedom are transferred to the corresponding independent degrees of freedom.

9. "FRAME" Data Block

This data block defines the properties, locations and loadings associated with the general three-dimensional FRAME (beam) elements that exist in the model. Any two-dimensional beam or truss element or any three-dimensional truss element may be considered as a special case of this general element. Skip this data block if there are no FRAME elements in the model. Otherwise prepare data for Format Sections a through e as described below.

FORMAT

a. Separator

Provide one data line for the FRAME separator in the following form:

FRAME

b. FRAME Control Information

Provide one data line for the FRAME control information in the following form:

NM=npro NL=nbsl NSEC=nsec
X=$x_1, x_2, \ldots, x_{nld}$ Y=$y_1, y_2, \ldots, y_{nld}$ Z=$z_1, z_2, \ldots, z_{nld}$
T=$t_1, t_2, \ldots, t_{nld}$ P=$pr_1, pr_2, \ldots, pr_{nld}$

DESCRIPTION

Variable	Note	Default	Entry
j_1	(1)		First joint number
j_2		[j_1]	Last joint number
inc		[1]	Joint number increment
Temperature Specification			
t_1	(2)	[0]	Temperature value at joint j_1
t_2		[t_1]	Temperature value at joint j_2
Pressure Specification			
p_1	(3)	[0]	Pressure value at joint j_1
p_2		[p_1]	Pressure value at joint j_2
w	(3)		Unit weight of fluid (weight/unit volume)
z			Global Z-ordinate of fluid surface

NOTES

1. The parameters j_1, j_2 and inc define the following series of joint numbers:

$$j_1,\ j_1+inc,\ j_1+2inc,\ j_1+3inc,...$$

which continues until j_2 is reached.

The pressure and/or temperature specification may be input or generated in any order of the joints. Specifications may

be repeated. For repeated joints, the last specification is used.

The temperature and pressure values are assumed to be zero for joints for which no specifications are input.

2. The temperature value assigned to joint j_1 is t_1. The temperature value assigned to joint j_2 is t_2.

The temperatures assigned to the generated joints are obtained by linear interpolation between t_1 and t_2 to give a total of $(1 + (j_2 - j_1)/inc)$ equally spaced values.

3. The joint pressures may be specified using either the P identifier or the W identifier. Both identifiers may not exist on the same data line.

If the P option is used, the pressure value assigned to joint j_1 is p_1, and the pressure value assigned to joint j_2 is p_2.

The pressures assigned to the generated joints are obtained by linear interpolation between p_1 and p_2 to give a total of $(1 + (j_2 - j_1)/inc)$ equally spaced values.

If the W option is used, the joint pressures for the specified series of joints is assumed to be due to fluid pressure (such as hydrostatic pressure). The joints are assumed to be submerged in the fluid and the plane defining the surface of the fluid is assumed parallel to the global X-Y plane at a level corresponding to the global Z-ordinate, z. The pressure value at a particular joint, j, is given by

$$p_j = w\ (z - z_j)$$

where z_j is the Z-ordinate of the joint j.

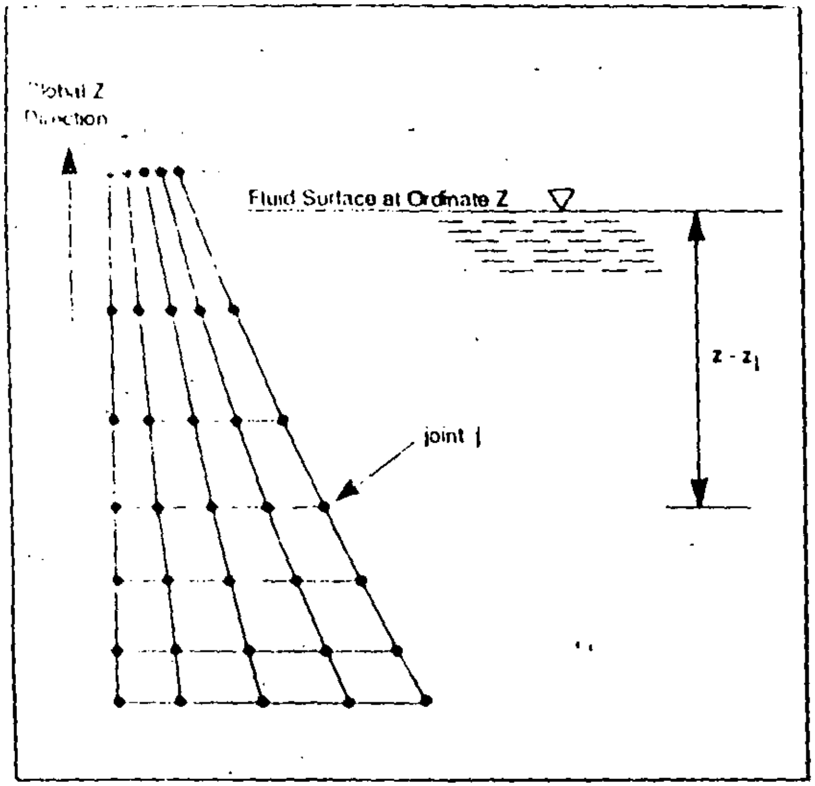


Figure X-7
Fluid Pressure Definition

If $(z - z_j)$ is negative, p_j is set to zero. See Figure X-7.

8. "CONSTRAINTS" Data Block

The data defined in this data block is used to reduce the number of equations in the system to be solved when the displacements along certain degrees of freedom are known to be equal to the displacements along other degrees of freedom. See Chapter IV, Section C. Skip this data block if there are no constraints to be specified. Otherwise prepare data Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the CONSTRAINTS separator in the following form:

CONSTRAINTS

b. Constraint Data

In this data section provide as many data lines as needed to define the joint constraints. **End this data section with a blank line.** Prepare the data lines in the following form:

j_1 j_2 inc C=c_{ux}, c_{uy}, c_{uz}, c_{rx}, c_{ry}, c_{rz}
I=i_{ux}, i_{uy}, i_{uz}, i_{rx}, i_{ry}, i_{rz}

EXAMPLE

CONSTRAINTS
21 31 1 C=20,0,0,0,0,0

7. "POTENTIAL" Data Block

This data block is for specifying joint temperature values and joint pressure values for a structural analysis.

The joint temperature values are used if a thermal analysis is required. Element properties and the corresponding joint temperature values are used to generate the element thermal load vectors. Alternatively, temperatures may be obtained automatically from a previous SAP90 Heat Transfer analysis [13].

The joint pressure values are used by the SHELL, ASOLID and SOLID elements for calculating pressure load vectors. Each element type interprets these joint pressure values differently.

- The SHELL element interprets the four pressure values corresponding to the four joints of the element as surface pressures in a direction normal to the plane of the element, and will generate a corresponding element load vector. The positive pressure convention associated with the SHELL element is shown in Figure X-17.

- The ASOLID and SOLID elements interpret the joint pressure values as scalar pressure quantities, defining a pressure gradient field through the volume of the structure. The element properties and corresponding joint values are used to generate the element pressure load vectors. Forces obtained from such pressure gradients are typically directed from regions of high pressure values toward regions of low pressure values.

This data block is only needed if a thermal or pressure analysis is required. If this data block is needed, prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the POTENTIAL separator in the following form:

POTENTIAL

b. Potential Data

In this data section provide as many data lines as needed to define the pressure and/or temperature fields acting on the structure. **End this data section with a blank line.** Prepare the data lines in the following form:

j1 j2 inc T=t1,t2 P=p1,p2 W=w,z

EXAMPLE

```
POTENTIAL
12 18 2 T=96,126 P=10,20
```

EXAMPLE

```
MASSES
5 25 5  M=1,1,0,0,0,1200
```

DESCRIPTION

Variable	Note	Default	Entry
j_1	(1)		First joint number
j_2		$[j_1]$	Last joint number
inc		[1]	Joint number increment
m_{ux}	(2,3)		X-translational mass
m_{uy}			Y-translational mass
m_{uz}			Z-translational mass
m_{rx}			Mass moment of inertia about the X-axis
m_{ry}			Mass moment of inertia about the Y-axis
m_{rz}			Mass moment of inertia about the Z-axis

NOTES

1. The parameters $\mathbf{j_1}$, $\mathbf{j_2}$ and **inc** define the following series of joint numbers:

   j_1, j_1+inc, j_1+2inc, j_1+3inc,...

   which continues until $\mathbf{j_2}$ is reached. All the joints in this series will receive the mass specification defined on this data line. The joints may be input in any order.

2. The degrees of freedom along which the masses are defined should not be restrained in the RESTRAINTS data block. Mass values must be in consistent mass units (W/g) and mass moments of inertia must be in (WL^2/g) units. Here W is weight, L is length, and g is the acceleration due to gravity.

   For repeated joints, the mass values along any degree of freedom are additive. *All directions are with respect to the global coordinate system.*

3. Joint masses prescribed in this data block are combined with the mass terms computed from the element mass densities.

5. "SPRINGS" Data Block

Any of the six degrees of freedom of any of the structural joints of the structure can have translational or rotational spring support conditions. This data block defines the locations of such spring supports and their associated values. Spring supports are not allowed along restrained degrees of freedom. Skip this data block if there are no spring constants to be defined in the model. Otherwise prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the SPRINGS separator in the following form:

SPRINGS

b. Spring Data

In this data section provide as many data lines as needed to define the spring supported degrees of freedom of the system. **End this data section with a blank line.** Prepare the data in the following form:

j_1 j_2 inc K=k_{ux}, k_{uy}, k_{uz}, k_{rx}, k_{ry}, k_{rz}

EXAMPLE

```
SPRINGS
5 25 5  K=1000,0,0,0,0,0
```


DESCRIPTION

Variable	Note	Default	Entry
j_1	(1)		First joint number
j_2		[j_1]	Last joint number
inc		[1]	Joint number increment
k_{ux}	(2)		X-translational spring constant
k_{uy}			Y-translational spring constant
k_{uz}			Z-translational spring constant
k_{rx}			X-rotational spring constant
k_{ry}			Y-rotational spring constant
k_{rz}			Z-rotational spring constant

NOTES

1. The parameters j_1, j_2 and inc define the following series of joint numbers:

   $$j_1, j_1+inc, j_1+2inc, j_1+3inc, ...$$

   which continues until j_2 is reached. All the joints in this series will receive the spring specification defined on this data line. The joints may be input in any order

2. The degrees of freedom along which the springs are applied should not be restrained in the RESTRAINTS data block. The translational spring stiffnesses k_{ux}, k_{uy} and k_{uz} must be entered with the units of force per unit of displacement.

The rotational spring stiffnesses $\mathbf{k_{rx}}$, $\mathbf{k_{ry}}$ and $\mathbf{k_{rz}}$ must be entered with the units of moment per unit radian of rotation. For repeated joints, the springs along any degree of freedom are additive. *All directions are with respect to the global coordinate system.*

6. "MASSES" Data Block

In a dynamic analysis mode, it may be necessary to lump concentrated nodal masses (and corresponding mass moments of inertia) at the joints. Any of the six degrees of freedom of any of the structural joints of the structure can have translational or rotational mass values. This data block defines the locations of such masses and their associated values. Information in this data block is only used by the program in a dynamic analysis mode. Mass values are not allowed along restrained degrees of freedom. Skip this data block if there are no mass values to be defined in the model. Otherwise prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the MASSES separator in the following form:

MASSES

b. Mass Data

In this data section provide as many data lines as needed to define the mass-loaded degrees of freedom of the system. **End this data section with a blank line.** Prepare the data in the following form:

j_1 j_2 inc M=m_{ux}, m_{uy}, m_{uz}, m_{rx}, m_{ry}, m_{rz}

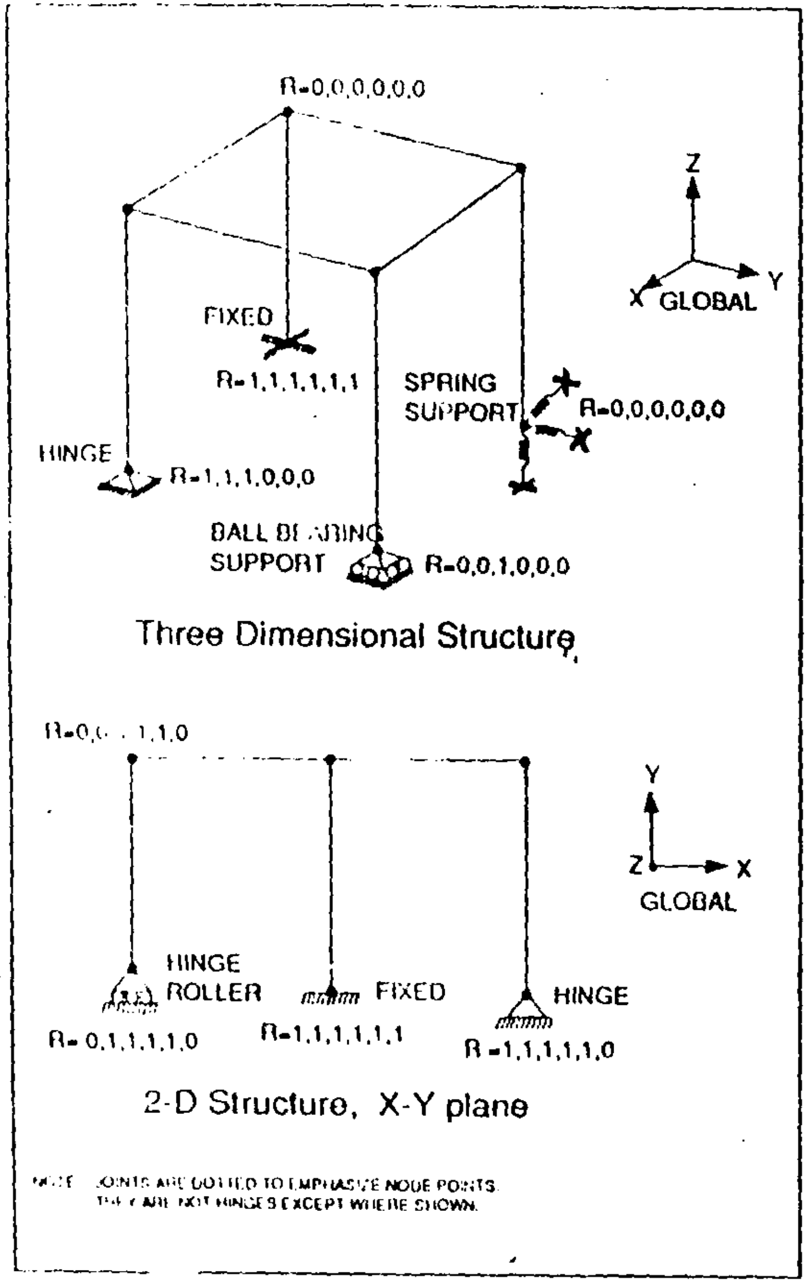


Figure X-6
Examples of Restraint Conditions

This data block is mandatory unless the model is completely supported by springs. Prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the RESTRAINTS separator in the following form:

RESTRAINTS

b. Restraint Data

In this data section provide as many data lines as needed to define inactive degrees of freedom of the system. **End this data section with a blank line.** Prepare the data in the following form:

j1 j2 inc R=r_{ux}, r_{uy}, r_{uz}, r_{rx}, r_{ry}, r_{rz}

EXAMPLE

RESTRAINTS
15 25 5 R=1,1,1,1,1,1

DESCRIPTION

Variable	Note	Default	Entry
j_1	(1)		First joint number
j_2		[j_1]	Last joint number
inc		[1]	Joint number increment
r_{ux}	(2)		X-translation restraint code
r_{uy}			Y-translation restraint code
r_{uz}			Z-translation restraint code
r_{rx}			X-rotation restraint code
r_{ry}			Y-rotation restraint code
r_{rz}			Z-rotation restraint code

NOTES

1. The parameters j_1, j_2 and inc define the following series of joint numbers:

   j_1, j_1+inc, j_1+2inc, j_1+3inc,...

   which continues until j_2 is reached. All the joints in this series will receive the restraint specification defined on this data line. The joints may be input in any order.

2. If the degree of freedom is inactive (restrained) enter 1, otherwise enter 0. Repeated joint specifications are allowed. However, degrees of freedom restrained (by 1) not be freed by subsequent respecifications (of 0), but

unrestrained degrees of freedom can be restrained by later specifications (of 1). It is recommended that all extraneous joints (joints with no members attached) be restrained with a restraint specification of R=1,1,1,1,1,1 in order to eliminate the associated degrees of freedom from the system.

No joint loads, springs or masses can be applied to restrained degrees of freedom.

The program will generate reactions at all restrained degrees of freedom.

All generated joints (joints within the four boundaries) satisfy the following conditions:

$$X_{ij} = (X_{i-1\,j} + X_{i+1\,j} + X_{i\,j-1} + X_{i\,j+1}) / 4$$

$$Y_{ij} = (Y_{i-1\,j} + Y_{i+1\,j} + Y_{i\,j-1} + Y_{i\,j+1}) / 4$$

$$Z_{ij} = (Z_{i-1\,j} + Z_{i+1\,j} + Z_{i\,j-1} + Z_{i\,j+1}) / 4$$

These equations are solved within the program by iteration. Since the coordinates are specified on all boundaries, convergence is obtained after a small number of iterations.

Joint 1 is located at the origin of the i-j system of joints that is generated and is the lowest joint number in the system. Joint numbers on the i-axis are incremented by 1. Joint numbers on the j-axis are incremented by $(n_i + 1)$.

7. c_1, c_2 and c_3 are previously defined joint numbers. The vector from c_1 to c_2 defines the positive direction of the reference axis. See Figure X-5.

Joint c_3 is on the generation circumference and must not lie on the c_1-c_2 line.

All generated coordinates are with reference to the joint c_3 and are located on a circular arc, with the reference axis as the center of the arc, at an incremental angle, a, between the radial lines associated with any two consecutive generated joints.

The positive direction along the arc is given by the right-hand rule with respect to the c_1-c_2 axis.

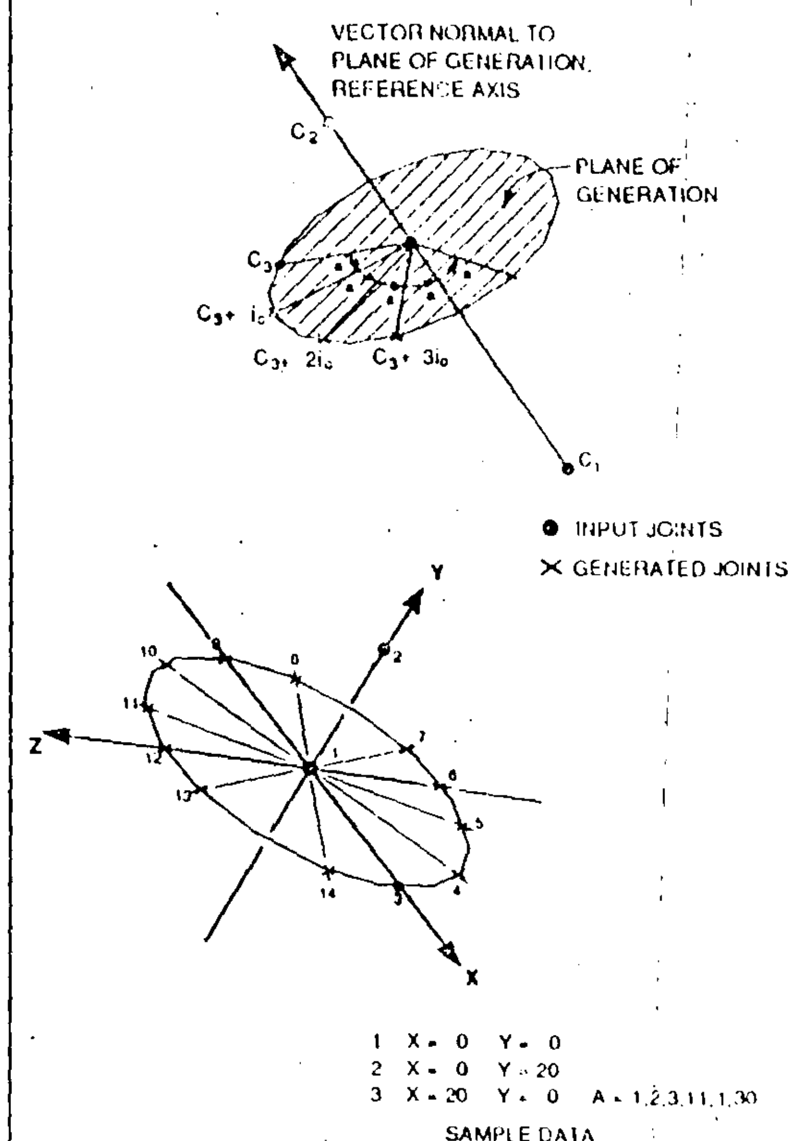


Figure X-5
Cylindrical and Spherical Generation

n_c additional joint numbers and coordinates are generated along the circular arc. The generated joint numbers are set to the previous joint number with a number increment of i_c.

Two additional parameters **h** and **r** can also be specified in the cylindrical generation allowing for spiral and helical generations. For spiral generation **h** is the height of the cylinder or cone and for helical generation **r** is the ratio of the radius of the last joint to the radius of the first joint in this generation. If these options are used the generation specification takes the form

$$A = c_1, c_2, c_3, n_c, i_c, a, h, r$$

and the default for **h** is 0.0 and for **r** is 1.0.

8. The generation identifiers G, Q, F, L and A may not be repeated on the same data line. Also, it should be noted if more than one generation is specified on any line, the program reads joint generation identifiers in the order of G, Q, F, L and A.

9. The X-, Y- and Z-ordinates are multiplied by the scale factor, s. If this parameter is not specified, it is set by the program to be unity (1). If s is set when the first joint is entered it need not be entered again, unless it is necessary to reset its value.

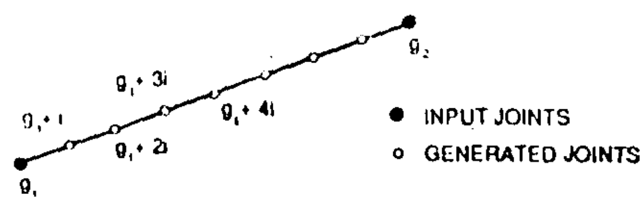
4. "RESTRAINTS" Data Block

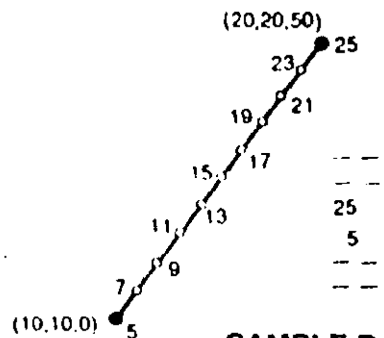
Every joint of the structural model has six displacement components, three global translations UX, UY and UZ, and three global rotations, RX, RY and RZ. The directions associated with these six displacement components are known as the **degrees of freedom** of the joint. See Chapter III, Section C.

The restraint specification of a joint consists of a set of six numbers (or codes), one number corresponding to each of the six degrees of freedom of the joint. Each of these numbers can have a value of 0 or 1.

If the displacement of a joint along any of its six degrees of freedom is known to be zero (e.g., support points) or if any degree of freedom of the joint is known to have no stiffness (e.g., extraneous joints), then the number corresponding to that degree of freedom in the restraint specification of that joint should be set to 1. Degrees of freedom that have a restraint specification of 1 are known as **inactive** or **null** degrees of freedom. Conversely, degrees of freedom that have a restraint specification of 0 are known as **active** degrees of freedom. See Figure X-6.

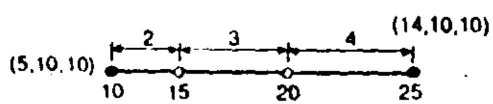
This data block identifies the support and null degrees of freedom of the structure. A joint with no inactive degrees of freedom need not be specified in this data block. Unrestrained degrees of freedom that have no stiffness are assigned a stiffness value of 1.0 by the program. This action eliminates the associated singularities from the system of equations and will obviously have no effect on the final results.





25 X = 20 Y = 20 Z = 50
5 X = 10 Y = 10 Z = 0 G = 5,25,2

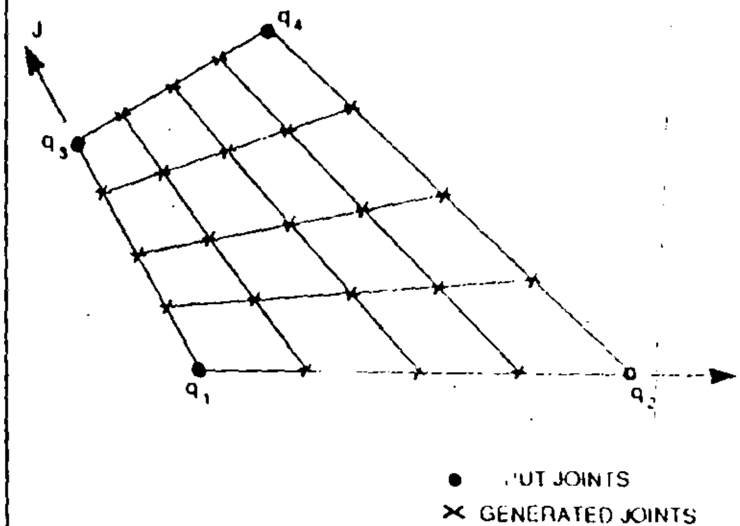
SAMPLE DATA
(EQUAL SPACES)

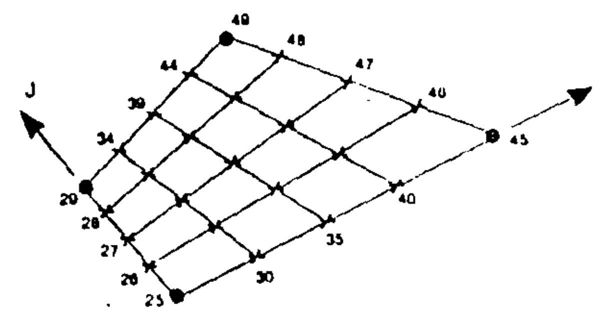


10 X = 5 Y = 10 Z = 10
25 X = 14 Y = 10 Z = 10 G = 10,25,5,2

SAMPLE DATA
(UNEQUAL SPACES)

Figure X-1
Linear Joint Generation





25 X = 10 Y = 20 Z = 0
45 X = 20 Y = 30 Z = 5
29 X = 15 Y = 30 Z = 10
49 X = 0 Y = 10 Z = 5 Q = 25,45,29,49,5,1

SAMPLE DATA

Figure X-2
Quadrilateral Joint Generation

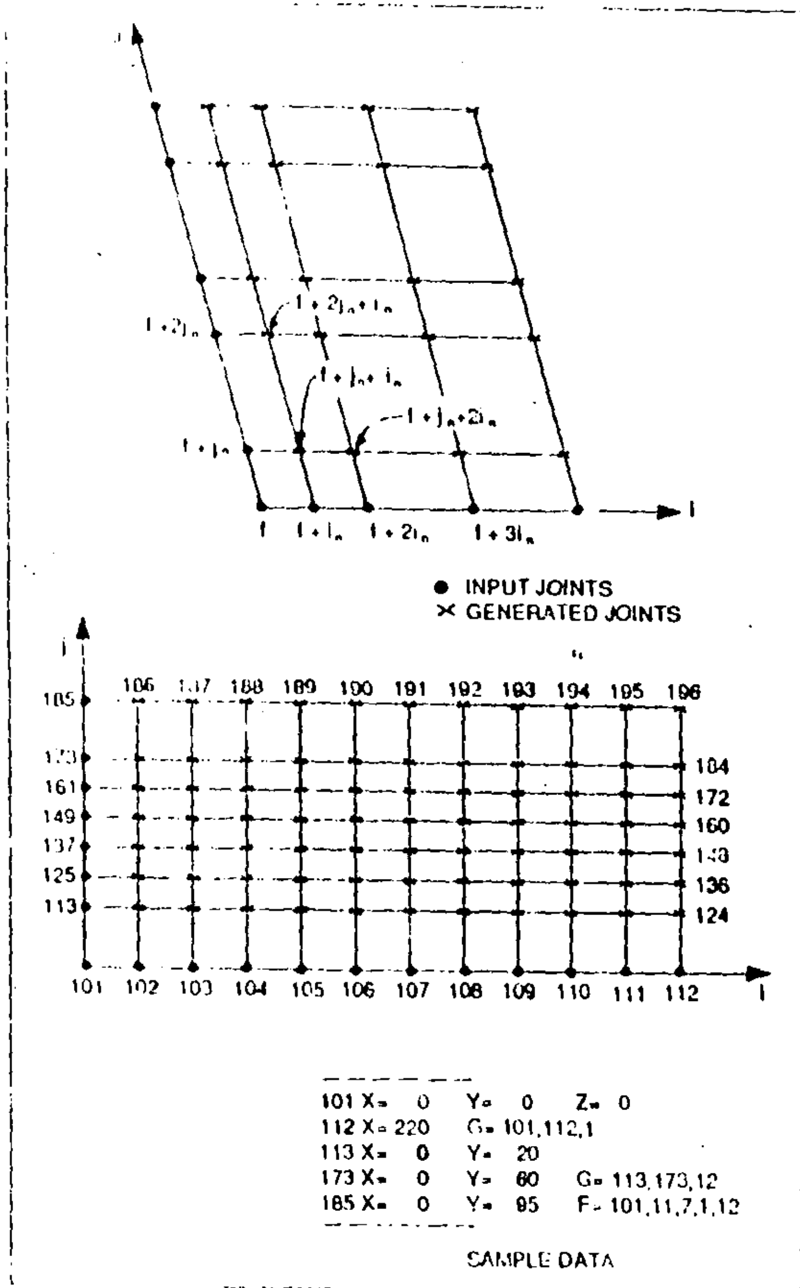


Figure X-3
Frontal Joint Generation

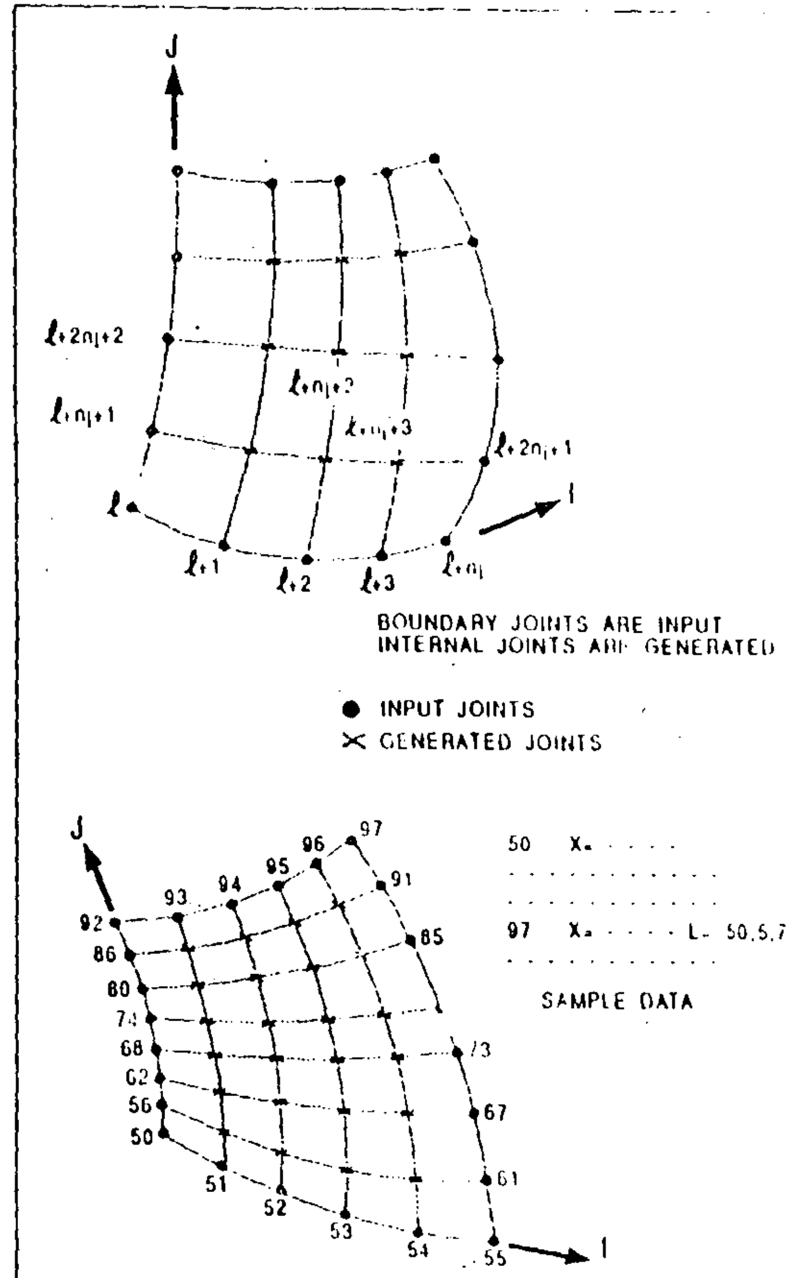


Figure X-4
Lagrangian Joint Generation

Variable	Note	Default	Entry
Scale Factor			
s	(9)	[pv]	Scale factor for joint c ordinates

NOTES

1. **jid** must be between 1 and **nid** (SYSTEM data block). Joint numbers do not have to be consecutive and may be supplied in any order. Joint coordinates may be re-specified or re-generated, in which case only the last definitions will be used. The final set of coordinates for all joints will be printed by the program.

   The user may define extraneous joints in the system that do not attach to any elements, if this helps the joint generation procedure. Restraining the six degrees of freedom of these extraneous joints is recommended, but not mandatory.

2. These coordinates are with respect to the global **X-Y-Z** coordinate system and are multiplied by the scale factor **s**. See Note 9 below.

   The g d Z-axis must point up if the SPEC data block is used, if the TIMEH data block is used with base-motion, or if rigid-floor diaphragms are modeled in the FRAME data block.

3. **g1** and **g2** are the joint numbers of two joints that have already been specified. The joint numbers that are generated will be equally or unequally (by arithmetic progression) spaced on the line that joins **g1** and **g2**, depending on the value of **r**. Joint number **g1** is successively incremented by **i** until **g2** is reached. This defines the number of joints that are generated on the line. See Figure X-1.

4. **q1**, **q2**, **q3** and **q4** are joint numbers of four joints that have been previously defined. These four joints define a quadrilateral: the direction **q1**-**q2** is an i-axis, and the direction **q1**-**q3** is a j-axis, as shown in Figure X-2.

   The joint numbers along the i-axis start at **q1**, which is successively incremented by **in** until **q2** is reached. This defines the number of joints along the i-axis.

   Similarly, the number of joints along the j-axis is defined by starting with **q1** and successively incrementing **q1** with **jn** until **q3** is reached.

5. **f** is a joint number associated with the frontal node generation option that allows for the automatic generation of joints along lines in three dimensions that are parallel to each other. As shown in Figure X-3, all the joints along the i-direction and the j-direction (the two sides of a parallelogram) have to be previously defined and must have joint numbers as shown in the figure. The joint numbers required to complete the parallelogram are generated by the program. The joint numbers of the generated joints are as shown in the figure.

   f is the joint number at the origin of the i-j system of joints being generated.

6. **l** is a joint number associated with the Lagrangian generation option that allows for the automatic generation of joints on a complex four-sided surface in space, given the coordinates of all the joints along the four sides, as shown in Figure X-4.

Only the actual number of modes that are evaluated will get included in any subsequent response spectrum or time-history analysis processing.

per can only be specified if **nfq** is specified.

7. If wopt is set to 1, all warning messages that are generated by the data check phase of the program (i.e., execution of the SAP90 command), will not appear in the echo output file (e.g. EXAMPLE.SAP).

   The messages, however, will always appear on the screen, irrespective of the value of **wopt**.

8. If **nritz** is specified, the program is put into Ritz-vector analysis mode.

   In this mode, the program calculates **nritz** orthogonal Ritz vectors. These vectors are load dependent as opposed to eigenvectors and the calculated time periods closely approximate the natural time periods. The Ritz-vector algorithm is more efficient than the eigenvector algorithm. For this analysis the program assumes the load to be base excitation. Ritz vectors, therefore, are recommended for all base-excitation problems as fewer of them will give a better approximation to the response.

9. nid specifies the largest identification number that may be used for the joints and for each of the four element types. Using a larger value of **nid** requires more memory. The value of **nid** may not exceed 32,767.

3. "JOINTS" Data Block

This data block defines the joints that describe the geometry of the structural model along with their associated coordinates. This data block is mandatory. Prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the JOINTS separator in the following form:

JOINTS

b. Joint Data

In this data section provide as many data lines as needed to define the joints in the structure. **End this data section with a blank line.** Prepare the data lines in the following form:

jid X=x Y=y Z=z G=g_1, g_2, i, r
Q=q_1, q_2, q_3, q_4, i_n, j_n F=f, n_i, n_j, i_n, j_n L=l, n_i, n_j
A=c_1, c_2, c_3, n_c, i_c, a S=s

EXAMPLE

```
JOINTS
25   X=20   Y=20   Z=20   S=12
5    X=10   Y=10   Z=0    G=5,25,2
```

DESCRIPTION

Variable	Note	Default	Entry
Joint Identification and Coordinates			
jid	(1)		Joint identification number
x	(2)	[pv]	Global X-ordinate of joint jid
y		[pv]	Global Y-ordinate of joint jid
z		[pv]	Global Z-ordinate of joint jid
Linear Generation			
g_1	(3,8)		Linear generation joint 1
g_2			Linear generation joint 2
i		[1]	Joint number increment
r		[1]	Ratio of last space to first space for unequal spacing of joints
Quadrilateral Generation			
q_1	(4,8)		Quadrilateral generation joint 1
q_2			Quadrilateral generation joint 2
q_3			Quadrilateral generation joint 3
q_4			Quadrilateral generation joint 4
i_n			Joint number increment, on i-axis
j_n			Joint number increment, on j-axis

Variable	Note	Default	Entry
Frontal and Lagrangian Generation			
f	(5,8)		Originating joint number for frontal generation
l	(6,8)		Originating joint number for Lagrangian generation
n_i			Number of joints in the i-direction, not including the originating joint number
n_j			Number of joints in the j-direction, not including the originating joint number
i_n			Joint number increment, on i-axis
j_n			Joint number increment, on j-axis
Cylindrical Generation			
c_1	(7,8)		Cylindrical generation joint 1
c_2			Cylindrical generation joint 2
c_3			Cylindrical generation joint 3
n_c			Number of additional cylindrical joints to be generated
i_c			Joint number increment for cylindrical joint generation
a			Increment angle, degrees (must be less than 90)

2. "SYSTEM" Data Block

This data block defines control information associated with the structural analysis. This data block is mandatory.

FORMAT

a. Separator

Provide one data line for the SYSTEM separator in the following form:

SYSTEM

b. Control Information

Following the SYSTEM separator provide one data line in the following form:

R=ropt L=nld C=ncyc V=nfq T=tol
P=per W=wopt Z=nritz N=nid

EXAMPLE

SYSTEM
V=10 L=3

DESCRIPTION

Variable	Note	Default	Entry
ropt	(1)	[0]	Restart flag: = 0 Normal execution mode = 1 Restart mode = 2 Restart after heat transfer analysis
nld	(2)	[0]	Number of load conditions
ncyc	(3)	[0]	Load frequency for steady-state analysis (cycles/time units)
nfq	(4)	[0]	Number of eigenvalues to be calculated
tol	(5)	[.0001]	Convergence tolerance for eigen-analysis
per	(6)	[0]	Cutoff time period for eigen-analysis
wopt	(7)	[0]	Warning suppression flag: = 0 Output all warnings = 1 Suppress all warnings
nritz	(8)	[0]	Number of Ritz vectors to be calculated
nid	(9)	[9999]	Maximum joint or element identification number

NOTES

1. This option allows the user to obtain output for additional loading combinations, activate or modify selective printing of the analysis output or perform the dynamic analysis of the structure for different time-history or response spec

trum data, without having to rerun the program through equation-solution and eigen-solution phases. Normally, **ropt** is set to 0; however, if **ropt** equals 1, all other entries on this data line are ignored, and are assumed to be the same as in the original run. See Chapter IX for details of the restart option.

2. In a static analysis mode, **nld** is the number of basic independent load conditions that act on the structure. Load combinations and envelopes are created as linear combinations of these load conditions. The program is assumed to in a static analysis mode if **ncyc** and **nfq** or **nritz** are not specified.

   In a steady-state analysis mode, **nld** is the number of spatial harmonic load distributions. The program is put into a harmonic steady-state analysis mode if **ncyc** is specified.

   In a time-history analysis mode, **nld** is the number of spatial load distributions, each of which may also be multiplied by a time-dependent function. The time-dependent function multipliers are specified in the TIMEH data block.

3. If **ncyc** is specified, the program is put into a harmonic steady-state analysis mode.

   In this mode, all specified joint and element loading is assumed to have a harmonic variation with a frequency of **ncyc**, the specified loading being the amplitude. The structural damping is assumed to be zero.

   It is possible to have **nld** spatial distributions, corresponding to the **nld** loading conditions that can be defined. If **ncyc** is specified, **nfq** or **nritz** must not be specified.

4. If **nfq** is specified, the program is put into an eigen-analysis mode. In this mode, **nfq** is the number of the lowest natural frequencies and the corresponding mode shapes that will be calculated in the eigenvalue analysis, and subsequently be included in the dynamic response-spectrum or time-history analysis if the SPEC or TIMEH data blocks are defined. The number of frequencies actually calculated is subject to the value of **per**. See Note 7 below.

   If **nfq** is specified, **ncyc** or **nritz** must not be specified.

   A static analysis and an eigen-analysis (with subsequent response-spectrum or time-history analysis) may be performed in the same run, i.e., **nld** and **nfq** can exist concurrently in the SYSTEM data.

   However, a harmonic analysis (with **nld** spatial distributions) and an eigen-analysis may not be performed in the same run, i.e., **ncyc** and **nfq** or **nritz** cannot exist concurrently in the SYSTEM data.

5. The eigenvalues are evaluated by an accelerated subspace iteration algorithm. The iteration will continue until the change in the time period T of a particular mode in successive iterations is less than **tol**, that is:

   $$| T(n) - T(n+1) | < tol$$

   where n and n+1 denote successive iteration numbers.

   tol can only be specified if **nfq** is specified.

6. The program will terminate the eigen-solution for the **nfq** time periods when all of the modes having a time period greater than **per** have been found.

- TIMEH
- COMBO
- ENVELOPE
- SELECT

This allows the user to obtain output for additional or modified dynamic excitation, additional or modified loading combinations and additional or modified lists of selected nodes or elements. All other data is assumed to be the same as in the original run.

The name of the input data file used for the restart run must be the same as the name of the input file for the original run. If desired, the original input file may be saved unde different filename using the MS-DOS **COPY** or **RENAM** ommand.

In the restart mode, the SAP90 module will only read the above mentioned six data blocks. Any other data blocks existing in the input data file are not required and will be ignored by the program if they exist.

All intermediate execution files created by the original run must be left on the disk if a restart run is anticipated.

As the name of the restart input data file must be the same as the name of the original data file, all output files (shown in Figure IX-1) associated with the original run will be overwritten by the restart run. If the output files associated with the original run are to be saved, the filenames should be changed using the MS-DOS **RENAME** command.

X.

DETAILS OF THE SAP90 INPUT DATA OPTIONS

This chapter describes the input formats needed to prepare a SAP90 data file. As described in Chapter VII, the SAP90 input data file is organized into twenty-one distinct **data blocks**. It is imperative that the user read the preceding chapters of this manual before attempting to prepare any data.

Only data associated with data blocks that are pertinent to the structure being analyzed need be prepared. For example, if the structure has no spring supports, the user skips the SPRING data block completely (including the separator line).

Similarly, if the model consists of only FRAME (beam) elements, the user will not provide any data associated with the SHELL, ASOLID or SOLID element data blocks.

Each data block is subdivided into one or more **data sections**, and the data sections are comprised of one or more **data lines**. In some cases, the data lines in a data section are grouped into multiple **data sets**. If any list-directed input option on any data line is not needed for a particular structure, the user simply skips the associated data list. The user does not have to prepare such data lists with zero or null entries unless needed to reset previous entries.

Typically, the manner in which the various input data blocks are described throughout this chapter is as follows. First, the **Format** specifications for each of the data lines making up the data sections are presented; in some instances, an **Example** of the format is given. This is followed by a **Description** of each of the user entry options (or variables) available on the data lines. Where needed, references are made to **Notes**. These notes describe the variables in more detail and give information to aid in the better understanding of the various options.

In the format specifications, bold-faced items indicate variable names which the user should replace with specific values appropriate to the problem being analyzed. Items not shown in bold face should be entered literally into the data file as shown in the format specifications. The format specification for a data line may be shown here as several lines of text. However, it should be entered as a single data line in the input data file, with continuation lines as necessary. See Chapter VII for more detail on the format and continuation of input data lines.

In certain cases, the program will assign values for the variables if they are not specified by the user. These default values, if applicable, are tabulated with each user entry option. A default value shown as "[pv]" (previous value) indicates that the value of the variable on the current data line is set equal to what it was on the previous line in the data section. In such cases the first data line in the data section must always define the variable.

Prepare data for each block (as required by the problem) according to the specifications of Blocks 1 through 21 presented below.

The use of the preprocessor SAPIN [4] may facilitate the preparation of the input data file.

1. The Title Line

Prepare one data line of up to 70 characters for output labeling. This information will appear on every page of the output files created by SAP90. This line *must* be the first line in the input data file. No comment lines may precede it.

This data block consists of only one data line and has no separator. This data line is always mandatory whether the run is a normal execution or a restart execution.

C. Sequence of Execution

The SAP90 Structural Analysis program requires that the analytical operations be performed in a rigorously predetermined sequence. The order of execution for the SAP90 modules is illustrated in Figure IX-2.

The SAP90 module is always executed first. An error-free execution of this module will create the GO.BAT file. For a normal execution (as opposed to a restart execution) the GO.BAT file will contain some or all of the following commands in the following sequence:

OPTIMIZE
FRAME
SHELL
ASOLID
SOLID
SOLVE
EIGEN
RITZ
SPEC
TIMEH
JOINTF
ELEMF
FRAMEF
SHELLF
ASOLIDF
SOLIDF

For a restart execution mode, the GO.BAT file will contain some or all of the following commands:

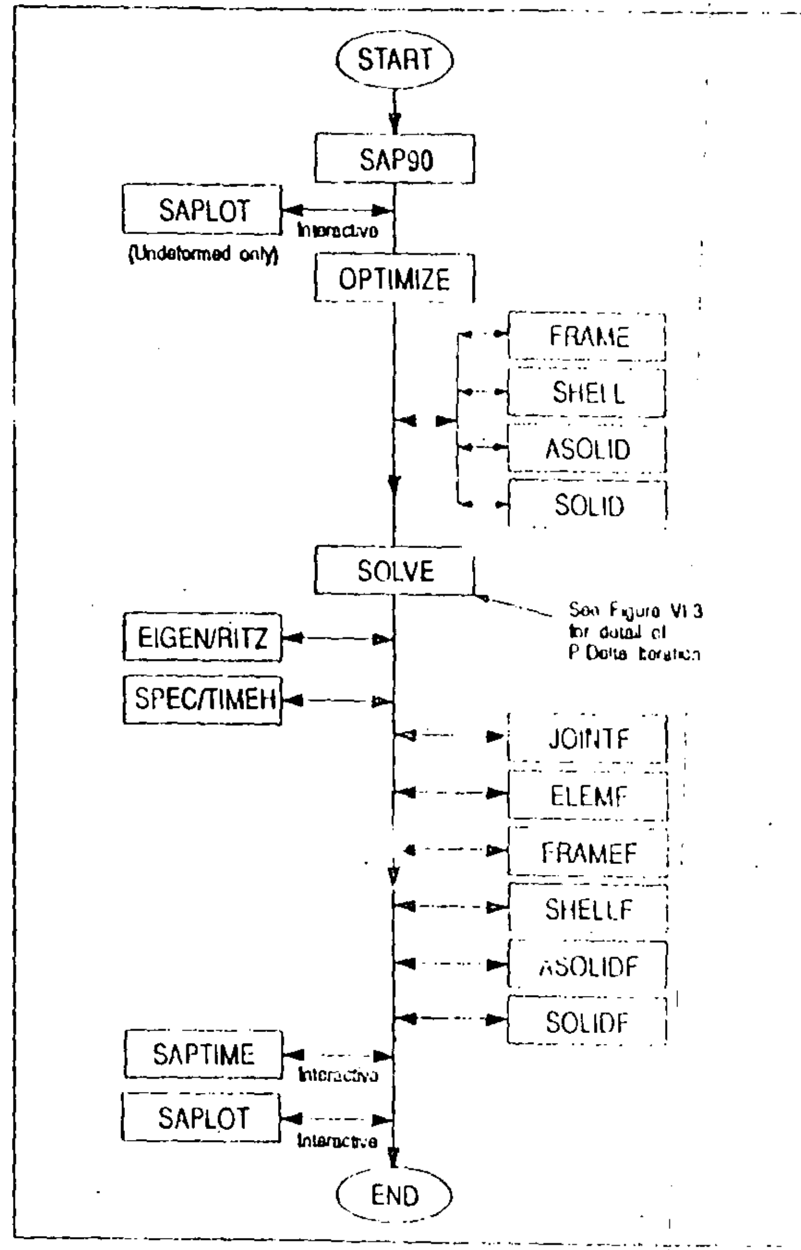


Figure IX-2
The SAP90 Program Structure
(without P-Delta)

SPEC
TIMEH
JOINTF
ELEMF
FRAMEF
SHELLF
ASOLIDF
SOLIDF

Note: each command in the GO.BAT file activates the corresponding program module.

The FRAME and FRAMEF commands will only exist if the model has FRAME elements. Similarly, the SHELL and SHELLF commands will only exist if the input contains the SHELL element data block. Similarly for the ASOLID, ASOLIDF and the SOLID, SOLIDF commands as they pertain to the ASOLID and SOLID elements.

The OPTIMIZE and SOLVE commands will always exist.

The EIGEN or RITZ commands will only exist if an eigenanalysis or Ritz-vector analysis has been activated.

The SPEC or TIMEH commands will only exist if a response-spectrum analysis or time-history analysis is being implemented, i.e., the SPEC data block or the TIMEH data block exists in the input data file.

The JOINTF command will always exist.

The ELEMF, FRAMEF, SHELLF, ASOLIDF and SOLIDF commands will only exist if loads are present or if the SPEC data block exists in the data file.

This order is altered when the P-Delta analysis option is activated. See Chapter VI, Section C.1 for more information. Additional commands may be present for a Bridge Analysis [12] or Heat Transfer Analysis [13].

The interactive graphic display module SAPLOT may be activated after an error-free execution of the SAP90 command to obtain graphic displays of the undeformed geometry of the model and the loading. Static deformed shapes, mode shapes, FRAME member-force diagrams and SHELL, ASOLID and SOLID element-stress contours may be obtained after the successful execution of the GO command.

The interactive module SAPTIME may be activated after the successful execution of the GO command for producing graphic displays and printed output for time-history analyses.

D. The Restart Option

The restart option of SAP90 allows the user to change certain data blocks, and to obtain output associated with the changed input data without involving the time-consuming assembly and solution phases of the program.

In a restart analysis mode, only the following data blocks may be changed or added:

- Title Line (Mandatory)
- SYSTEM (Mandatory)
- SPEC

IX.

SAP90 PROGRAM STRUCTURE AND RESTART

The purpose of this chapter is to outline the overall organization of the SAP90 Structural Analysis program.

This information is included to promote a better understanding of the execution sequence of the program. Details associated with the restart option of the program are also presented. This chapter should be read only after the user has acquired experience using the program as described in Chapter II. First-time users should skip this chapter.

A. The SAP90 Modules

The SAP90 Structural Analysis program is actually a combination of program modules that are executed in a predefined sequence. The modules are linked through a series of internal files that constitute a data base. Each module performs a series of operations and updates the data base in the process. The internal files have the same name as the input data file, but different file extensions.

The program modules included in the complete package of this release of SAP90, and the associated function of each module

PROGRAM MODULE	FUNCTION OF MODULE	OUTPUT FILE CREATED
1. SAP90	Read, Check and Tabulate all Input Data	EXAMPLE.SAP
2. OPTIMIZE	Optimize Equation Numbers	EXAMPLE.EQN
3. FRAME	FRAME Element Stiffness Formulation	
4. SHELL	SHELL Element Stiffness Formulation	
5. ASOLID	ASOLID Element Stiffness Formulation	
6. SOLID	SOLID Element Stiffness Formulation	
7. SOLVE	Stiffness and Load Blocking, Assembly and Reduction	
8. EIGEN	Eigenvalue Analysis	EXAMPLE.EIG
9. RITZ	Ritz-vector Analysis	EXAMPLE.RIT
10. SPEC	Response-Spectrum Analysis	EXAMPLE.SPC
11. TIMEH	Time-History Analysis	
12. JOINTF	Modes, Joint Displacements and Reactions Output	EXAMPLE.SOL
13. ELEMF	Element Nodal Forces Output	EXAMPLE.FEF
14. FRAMEF	FRAME Element Forces Output	EXAMPLE.F3F
15. SHELLF	SHELL Element Forces Output	EXAMPLE.F4F
16. ASOLIDF	ASOLID Element Stresses Output	EXAMPLE.F5F
17. SOLIDF	SOLID Element Stresses Output	EXAMPLE.F8F
18. SAPLOT	Graphic Displays	
19. SAPTIME	Graphic Displays and Output Creation from Time-History Analysis	EXAMPLE.HST

Figure IX-1
The SAP90 Structural Analysis Modules

are listed in Figure IX-1. Some of the modules create output files. The names of the associated output files are also shown.

Additional program modules are used by SAP90 Bridge Analysis [12] and by SAP90 Heat Transfer Analysis [13]. These are described in the referenced User Manuals.

Note: the names of a few of the SAP program modules are the same as the names of the SAP input data blocks. Do not let this become a source of confusion.

B. The GO Command

The SAP90 module reads the input file and checks all of the data for compatibility. This module is activated by giving the command:

C> SAP90 <CR>

If no errors are encountered, the SAP90 module will create a batch file GO.BAT which will activate the other modules of SAP90 that are needed by the particular analysis. The contents of the GO.BAT file depend upon the input options that are activated in a particular SAP90 input data file.

Therefore, only the modules that are actually required for the analysis of a particular model will be executed.

The GO.BAT file is activated after the execution of SAP90 by giving the command:

C> GO <CR>

```
[illegible]  S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE  1
                                              PROGRAM:SAP90/FILE:example.F8F
EXAMPLE [illegible] 24    [illegible]ANTILEVER BEAM USING EACH TYPE OF ELEMENT

S T R E S S E S   A T   C E N T E R   O F   E L E M E N T

ELEMENT ID       1
[illegible]   SXX         SYY          SZZ          SXY          SXZ          SYZ
 1    [illegible]  [illegible]  [illegible]  .2478E+00  [illegible]  -.1389E+02
 2    [illegible]  .2951E+[illegible]  [illegible]  .1001E-01  .2492[illegible]  .6259E+00
 3    [illegible]  .1069E+03  -.2018E+01  .2746E+00  .9574E+01  -.1326E+02

[illegible]  S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE  2
                                              PROGRAM:SAP90/FILE:example.F8F
EXAMPLE   EXAMPLE 24   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

A V E R A G E   J O I N T   S T R E S S E S

LOAD COMBINATION    1

JOINT        SXX          SYY          SZZ          SXY          SXZ          SYZ
 13    [illegible]  .3287E+03  .1117E+02  -.5276E+00  .9052E+01  -.1389E+02
 14    [illegible]  [illegible]  .1602E+[illegible]  .5276E+00  .8793E+01  -.1385E+02

[illegible]  S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE  3
                                              PROGRAM:SAP90/FILE:example.F8F
EXAMPLE   EXAMPLE 24   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

A V E R A G E   J O I N T   S T R E S S E S

LOAD COMBINATION    2

JOINT        SXX          SYY          SZZ          SXY          SXZ          SYZ
 13    .8720E+[illegible]  .5485E+01  .6667E+00  .2130E-01  .7689E+00  .6259E+00
 14    .4503E+00  .5665E+01  .6661E+00  .2130E-01  .2285E+00  .6259E+00

[illegible]  S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE  4
                                              PROGRAM:SAP90/FILE:example.F8F
EXAMPLE   EXAMPLE 24   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

A V E R A G E   J O I N T   S T R E S S E S

LOAD COMBINATION    3

JOINT        SXX          SYY          SZZ          SXY          SXZ          SYZ
 13    .3272E+02  .2030E+03  .1130E+03  -.5063E+03  .1013E+02  -.1326E+02
 14    .2488E+02  -.2030E+03  .1468E+02  -.5063E+00  .9726E+01  -.1326E+02

[illegible]  S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE  5
                                              PROGRAM:SAP90/FILE:example.F8F
EXAMPLE   EXAMPLE 24   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

P R I N C I P A L   J O I N T   S T R E S S E S

LOAD COMBINATION    1

JOINT     S11       S22       S33   V1X  V1Y  V1Z  V2X  V2Y  V2Z  V3X  V3Y  V3Z
 13    [illegible]  [illegible]  [illegible]  .00 1.00 .07 [illegible] .03 -.37 .37 -.06 .92
 14    [illegible]  [illegible]  .19E+02  .03 1.00 -.06 -.90 -.01 .20 -.19 .06 -.96
```

Figure VIII-17
Typical SOLID Element Output File "EXAMPLE.F8F"

8.11. File EXAMPLE.FEF

This file contains the element joint forces for all types of elements.

The element results are for each of the static (or steady-state) load conditions and the dynamic load condition, depending upon the options that are activated. If load combinations are requested, the results are for the load combinations and not for the load conditions.

The element joint forces are obtained by multiplying the element displacements and element stiffness matrices.

The element joint forces are output in the global coordinate system and are forces acting on the element at the joints and must be in equilibrium with the body forces (span loads, self weight, pressure, temperature, etc.) for all static load conditions.

Typical output is presented in Figure VIII-18.

```
                          FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  1
                                            PROGRAM SAP90/FILE example.FEF
       EXAMPLE 24  CANTILEVER BEAM USING EACH TYPE OF ELEMENT

   F R A M E     E L E M E N T   J O I N T   F O R C E S

 ELEMENT ID       1
  LOAD
  COND  JOINT      F(X)        F(Y)        F(Z)        M(X)        M(Y)        M(Z)
   1
          55   [illegible]  -.982E 08    .102E+04   -.720E+05    .193E+04    .492E-09
          55   [illegible]   .982E 08    .100E+04    .640E+05    .193E+04    .130E-09
   2
          55        .617    126.521      68.399    3192.891      85.725       6.431
          57        .617    126.521      68.399    3982.144      85.725       1.692
   3
          55   [illegible]   .127E+03    .103E+04   -.688E+05   -.184E+04    .643E+01
          57   [illegible]   .127E+03    .932E+03    .800E+05    .201E+04    .169E+01

 ELEMENT ID       2
  LOAD
  COND  JOINT      F(X)        F(Y)        F(Z)        M(X)        M(Y)        M(Z)
   1
           4   [illegible]   .982E 08    .102E+04   -.720E+05    .193E+04    .315E-09
          56   [illegible]   .982E 08    .100E+04    .640E+05   -.193E+04    .152E-08
   2
          56       2.374    126.521      68.399    3192.891      85.725       5.066
          36       2.374    126.521      68.399    3982.144      85.725      19.819
   3
          34   [illegible]   .127E+03    .107E+04   -.688E+05    .201E+04    .507E+01
          36   [illegible]   .127E+03   -.932E+03    .800E+05   -.184E+04    .198E+02

                          FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  2
                                            PROGRAM SAP90 FILE example.FEF
EXAMPLE    EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 S H E L L     E L E M E N T   J O I N T   F O R C E S

 ELEMENT ID       1
  LOAD
  COND  JOINT      F(X)        F(Y)        F(Z)        M(X)        M(Y)        M(Z)
   1
          22    .102E 08   -.939E 08    .100E+04   -.480E+05    .822E+04   -.818E-09
          31    .129E 08    .103E 07   -.100E+04    .600E+05    .871E+03   -.406E-09
          25    .278E 08    .103E 07    .100E+04   -.480E+05   -.822E+04   -.101E-08
          32    .124E 09    .993E 08   -.100E+04    .600E+05   -.871E+03   -.595E-09
   2
          22       9.615    102.477      61.724    1719.505     297.470       9.691
          31      11.002    123.517      61.724    2431.281      26.311       4.530
          25      21.547    112.921      61.724    1719.505     297.470      11.273
          32        .126    101.827      61.724    2431.287      26.311       4.039
   3
          22    .981E+01    .102E+03    .104E+04   -.463E+05    .852E+04    .969E+01
          31    .132E+02    .114E+03    .930E+03    .624E+05    .835E+03    .453E+01
          25    .215E+02    .113E+03    .106E+04   -.463E+05   -.792E+04    .113E+02
          32    .126E+00    .102E+03   -.910E+03    .624E+05   -.847E+03    .604E+01
```

(Output truncated for brevity.)

Figure VIII-18

Typical Element Joint Force Output File "EXAMPLE.FEF"

B.12. File EXAMPLE.ERR

This file contains errors and warnings that are generated by the program during the execution of the solution phases of the program.

Reasons for abnormal termination of a program during solution, such as excessive loss of solution accuracy, convergence failure in the eigensolution, etc., are reported in this file.

This file is empty for the present example since no errors or warnings were generated.

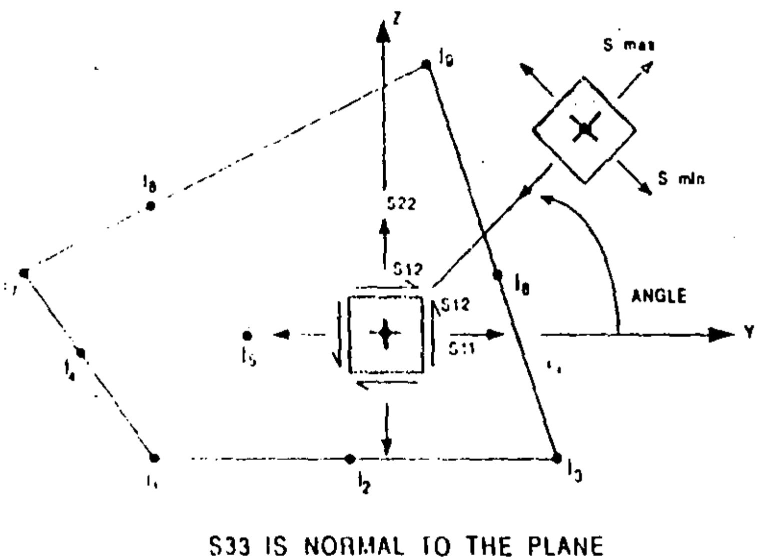


Figure VIII-14
ASOLID Element Stresses
(for Elements in Y-Z Plane)

```
C S I  /  S A P 9 0  -  -  FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE    1
                                          PROGRAM SAP90/FILE example F5F
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT -

A S O L I D   E L E M E N T   S T R E S S E S

ELEMENT ID      1 ----------------------------
LOAD COMBO      1 ----------------------------
JOINT       S11      S22      S33      S12    S(MAX)   S(MIN)    ANGLE
    1     20.16     -.14      .00   -20.82    33.17    13.15    32.02
    2     83.30     -.03      .00   -20.99    88.29    -5.02    13.37
    3    146.44      .08      .00    20.94   149.38     2.86     7.99
    6       .00      .00      .00   -36.60    36.60   -36.60    45.00
    7       .00      .00      .00   -36.77    36.77   -36.77    45.00
    8       .00      .00      .00    36.33    36.33    36.33    45.00
   11    -20.16      .14      .00   -20.82    13.15    33.17    58.00
   12    -83.30      .03      .00   -20.99     5.02    88.29   -76.63
   13   -146.44     -.08      .00   -20.94     2.86   149.38    82.01

ELEMENT ID      1 ----------------------------
LOAD COMBO      2 ----------------------------
JOINT       S11      S22      S33      S12    S(MAX)   S(MIN)    ANGLE
    1       .16      .11      .00      .30
    2      1.46      .10      .00      .49
    3      2.91      .10      .00      .68
    6       .11      .04      .00      .62
    7       .42      .01      .00      .62
    8       .72      .01      .00     1.01
   11       .16      .11      .00      .30
   12      1.46      .10      .00      .49
   13      2.91      .10      .00      .68

ELEMENT ID      1 ----------------------------
LOAD COMBO      3 ----------------------------
JOINT       S11      S22      S33      S12    S(MAX)   S(MIN)    ANGLE
    1     20.33     -.03      .00   -20.51
    2     84.76      .07      .00   -20.50
    3    149.35      .18      .00   -20.27
    6       .11      .04      .00   -35.98
    7       .42      .01      .00    35.96
    8       .72      .01      .00    35.71
   11    -20.00      .25      .00   -20.51
   12    -81.84      .13      .00   -20.50
   13   -143.53      .02      .00   -20.27
```

Figure VIII-15
Typical ASOLID Element Output File "EXAMPLE.F5F"

B.10. File EXAMPLE.F8F

This file contains the element stress output for the SOLID element.

The element results are for each of the static (or steady-state) load conditions and the dynamic load condition, depending upon the options that are activated. If load combinations are requested, the results are for the load combinations and not for the load conditions.

All results correspond to the global coordinate system.

Direct stresses and shear stresses are output at the center of each element. In addition, the stresses obtained from the joints of each element connecting to a particular joint are averaged to create tables of average joint stresses.

Principal joint stresses and associated principal direction cosines are output for static load conditions only.

Due to the CQC and SRSS techniques, the element output from the dynamic load condition will have no signs.

The sign conventions for SOLID element output are defined in Figure VIII-16. Typical SOLID element output is presented in Figure VIII-17.

This file will only exist if SOLID elements are present in the model being analyzed.

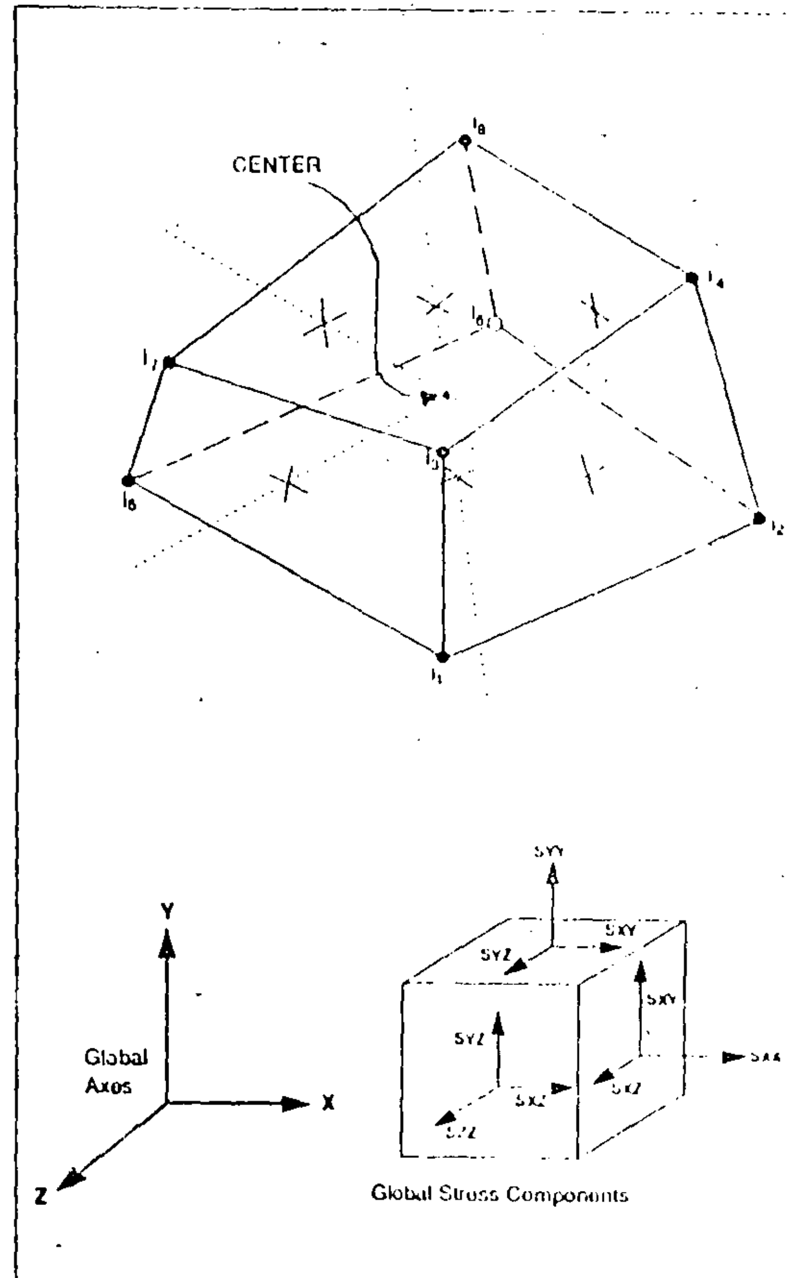


Figure VIII-16
SOLID Element Stresses

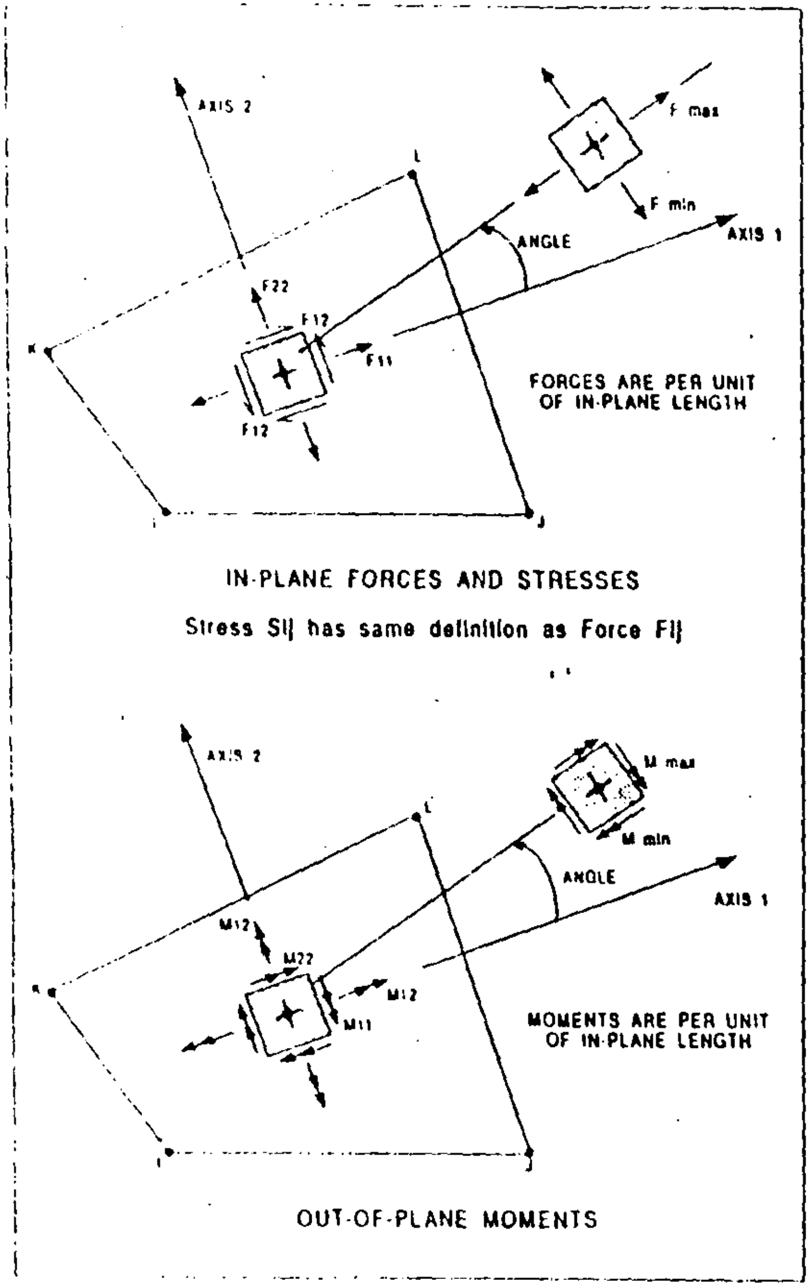


Figure VIII-11
SHELL Element Stresses and Resultants

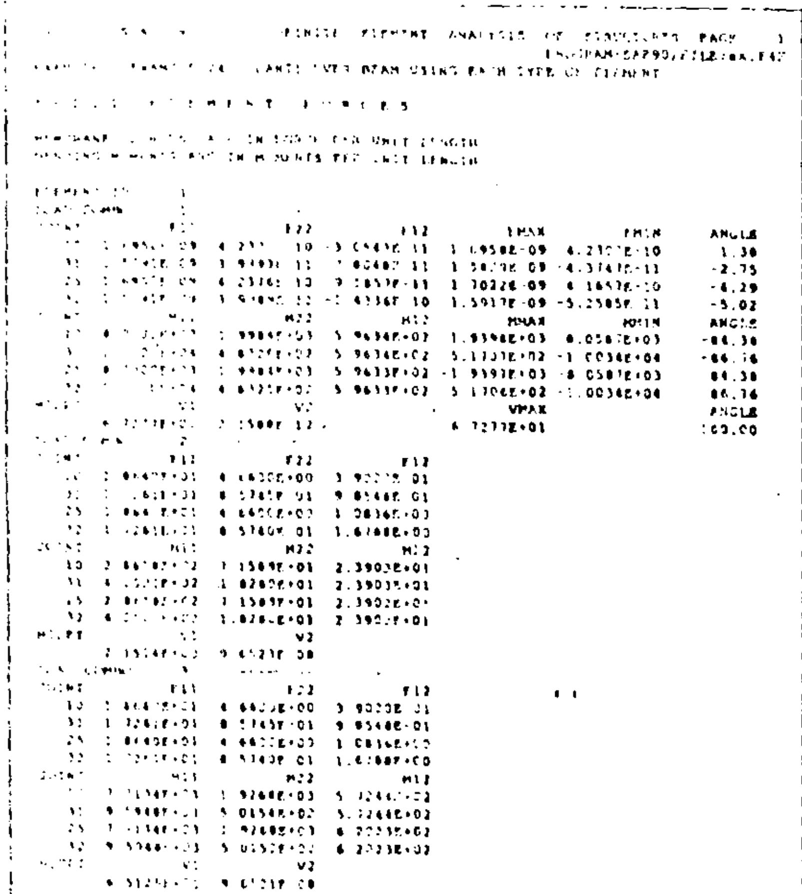

```
C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE
                                                PROGRAM SAP90 FILE example.[illegible]
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

S H E L L   E L E M E N T   S T R E S S E S

ELEMENT ID      1 ------------
LOAD COMBO      1 ------------
JOINT      S11 TOP     S22 TOP     S12 TOP    SMAX TOP    SMIN TOP     ANGLE
   10  3.3333E+02  8.3260E+01  2.4847E+01  3.3570E+02  8.0823E+01      5.62
   31  4.1667E+02 -2.0136E+01  2.4847E+01  4.1808E+02 -2.1545E+01      3.25
   25  3.3333E+02  8.3260E+01 -2.4847E+01  3.3570E+02  8.0823E+01      5.62
   32  4.1667E+02  2.0135E+01 -2.4847E+01  4.1808E+02 -2.1544E+01      3.25
JOINT      S11 BOT     S22 BOT     S12 BOT    SMAX BOT    SMIN BOT     ANGLE
   10 -3.3333E+02 [illegible]  -2.4847E+01 -8.0823E+01 -3.3570E+02    -84.38
   31 -4.1667E+02  2.0136E+01 -2.4847E+01  2.1545E+01 -4.1808E+02    -86.76
   25 -3.3333E+02 -8.3260E+01  2.4847E+01 -8.0823E+01  3.3570E+02     84.38
   32 -4.1667E+02  2.0135E+01  2.4847E+01  2.1544E+01  4.1808E+02     86.76
LOAD COMBO      2 ------------
JOINT      S11 TOP     S22 TOP     S12 TOP
   10  1.2042E+01  3.0081E+00  9.9647E-01
   31  1.6945E+01  7.6501E-01  9.9932E-01
   25  1.2042E+01  3.0080E+00  1.0000E+00
   32  1.6945E+01  7.6499E-01  1.0057E+00
JOINT      S11 BOT     S22 BOT     S12 BOT
   10  1.2042E+01  3.0081E+00  9.9647E-01
   31  1.6945E+01  7.6501E-01  9.9937E-01
   25  1.2042E+01  3.0080E+00  1.0000E+00
   32  1.6945E+01  7.6499E-01  1.0057E+00
[illegible] COMBO      3 ------------
JOINT      S11 TOP     S22 TOP     S12 TOP
   10  3.4537E+02  8.6274E+01  2.5844E+01
   31  4.3361E+02 -1.9371E+01  2.5847E+01
   25  3.4538E+02  8.6276E+01 -2.3847E+01
   32  4.3361E+02 -1.9370E+01 -2.3841E+01
JOINT      S11 BOT     S22 BOT     S12 BOT
   10 -3.2129E+02 -8.0260E+01 -2.3851E+01
   31 -3.9972E+02  2.0901E+01 -2.3848E+01
   25 -3.2129E+02 -8.0260E+01  2.5047E+01
   32 -3.9912E+02  2.0900E+01  2.5053E+01
```

(Output truncated for brevity.)

Figure VIII-12
Typical SHELL Element Output File "EXAMPLE F4F"
Stress Output as Requested in Input Data File



(Output truncated for brevity.)

Figure VIII 13
Alternative SHELL Element Output File "EXAMPLE.F4F" Force and Moment Resultants (Default Output)

B.9. File EXAMPLE.F5F

This file contains the element stress output for the ASOLID element.

The element results are for each of the static (or steady-state) load conditions and the dynamic load condition, depending upon the options that are activated. If load combinations are requested, the results are for the load combinations and not for the load conditions.

All results correspond to the global coordinate system.

Direct stresses and shear stresses are output for each joint of the element.

Principal stresses and associated principal directions are output for static load conditions only.

Due to the CQC and SRSS techniques, the element output from the dynamic load condition will have no signs.

The sign conventions for ASOLID element output are defined in Figure VIII-14. Typical ASOLID element output is presented in Figure VIII-15.

This file will only exist if ASOLID elements are present in the model being analyzed.

B.7. File EXAMPLE.F3F

This file contains the element force output for the FRAME element.

The element results are for each of the static (or steady-state) load conditions and the dynamic load condition, depending upon the options that are activated. If load combinations are requested, the results are for the load combinations and not for the load conditions. If envelopes are requested, the results are for the envelopes instead of for the load conditions or load combinations.

All results correspond to the element local coordinate system. Section forces (moments and shears) are produced at the face of the supports as shown on Figure X-16. Section forces are also produced for other controlling locations along the element length for span loaded elements. Optionally, the user can specify the number of equally spaced sections along the clear length of the member where section forces are to be produced.

The axial force and torque are always output at the two ends of the full length of the element.

The location of the output points is identified by the distance of the point from the joint at End I of the element.

Due to the CQC and SRSS techniques, the element output from the dynamic load condition will have no signs.

The sign conventions for FRAME element output are defined in Figure VIII-9. Typical FRAME element output is presented in Figure VIII-10.

This file will only exist if FRAME elements are present in the model being analyzed.

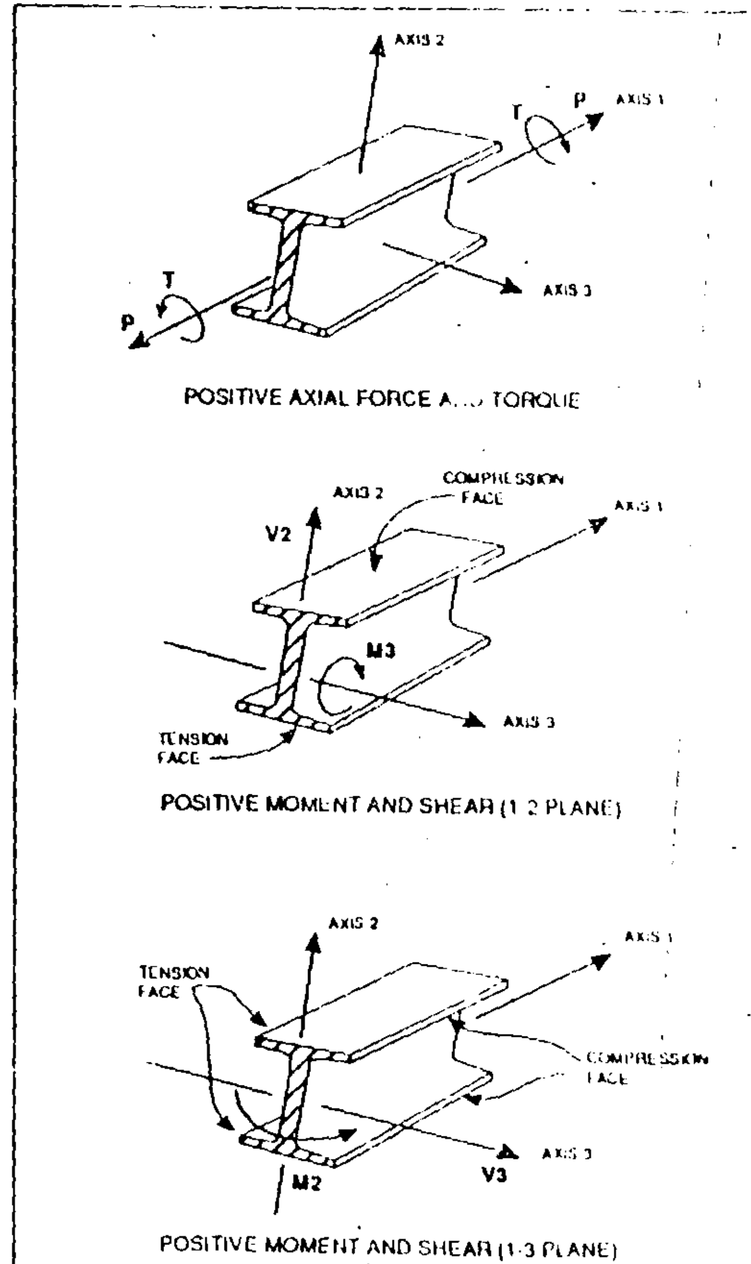


Figure VIII-9
FRAME Element Forces

SAP90 FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 1
PROGRAM:SAP90/FILE:EXAMPLE.F3F
EXAMPLE EXAMPLE 2A CANTILEVER BEAM USING EACH TYPE OF ELEMENT

FRAME ELEMENT FORCES

ELEMENT ID	[illegible]	1-2 PLANE SHEAR	1-2 PLANE MOMENT	AXIAL FORCE	1-3 PLANE SHEAR	1-3 PLANE MOMENT	AXIAL TORQ
1	.000			.000			1928.513
	.000	1000.000	72000.000		.000	.000	
	12.000	1000.000	84000.000		.000	.000	
	12.000			.000			1928.513
2	.000			126.521			85.725
	.000	68.399	3192.891		.477	6.431	
	12.000	68.399	3982.144		.477	1.692	
	12.000			126.521			85.725
3	.000			126.521			2016.238
	.000	1068.399	75192.891		.477	6.431	
	12.000	1068.399	87982.144		.477	1.692	
	12.000			126.521			2016.238
1	.000			.000			-1928.513
	.000	1000.000	72000.000		.000	.000	
	12.000	1000.000	84000.000		.000	.000	
	12.000			.000			-1928.513
2	.000			126.521			85.725
	.000	68.399	3192.891		2.074	5.066	
	12.000	68.399	3982.144		2.074	19.819	
	12.000			126.521			85.725
3	.000			126.521			-1842.788
	.000	1068.399	75192.891		2.074	5.066	
	12.000	1068.399	87982.144		2.074	19.819	
	12.000			126.521			-1842.788

Figure VIII-10
Typical FRAME Element Output File "EXAMPLE.F3F"

B.8. File EXAMPLE.F4F

This file contains the element response output for the SHELL element. Either resultant forces and moments, or top and bottom surface stresses may be produced.

The element results are for each of the static (or steady-state) load conditions and the dynamic load condition, depending upon the options that are activated. If load combinations are requested, the results are for the load combinations and not for the load conditions.

All results correspond to the element local coordinate system.

Resultant output consists of: in-plane direct forces and shear forces and out-of-plane moments for each joint of the element, and out-of-plane shear forces at the center of the element. All resultants are given per unit of in-plane length.

Stress output consists of: in-plane direct stresses and shear stresses at the top and bottom surfaces for each joint of the element; no out-of-plane shear stresses are produced.

Principal values and associated principal directions are output for static load conditions only.

Due to the CQC and SRSS techniques, the element output from the dynamic load condition will have no signs.

The sign conventions for SHELL element output are defined in Figure VIII-11. Typical SHELL element stress output is presented in Figure VIII-12; typical resultant output is presented in Figure VIII-13;

This file will only exist if SHELL elements are present in the model being analyzed.

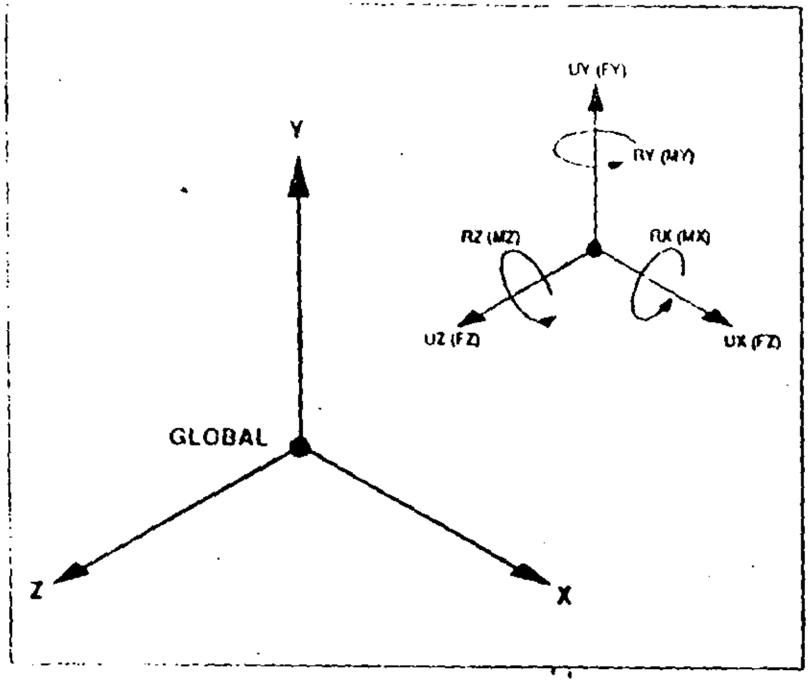


Figure VIII-7
Positive Joint Displacements and Reactions

The global vectors of reactions (and applied loads) are created by assembling the element joint reactions from each element. The element joint reactions are obtained by multiplying the element displacements and element stiffness matrices. A balance between the applied loads and the joint reactions, obtained from assembling the element joint loads, represents the necessary and sufficient condition for verification of the accuracy and stability of the solution.

C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 1
PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

M O D E S H A P E S

MODE SHAPE NUMBER 1 PERIOD = .031492 SECONDS

DISPLACEMENTS "U" AND ROTATIONS "R"

JOINT	U(X)	U(Y)	U(Z)	R(X)	R(Y)	R(Z)
11	.000000	.117068	1.197881	.000000	.000000	.000000
12	.000000	.116132	.963789	.000000	.000000	.000000
13	.000000	.113155	.752676	.000000	.000000	.000000
14	.000000	.105059	.512460	.017646	.000000	.000000
15	.000000	.090605	.313597	-.015100	.000000	.000000
31	.000000	.000000	.152687	-.011461	-.000648	.000000
33	.000000	.000000	.044117	-.006427	-.000563	.000000
35	.000000	.000000	.000000	.000000	.000000	.000000

C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 2
PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

M O D E S H A P E S

MODE SHAPE NUMBER 2 PERIOD = .005333 SECONDS

DISPLACEMENTS "U" AND ROTATIONS "R"

JOINT	U(X)	U(Y)	U(Z)	R(X)	R(Y)	R(Z)
11	.000000	.388371	1.119025	.000000	.000000	.000000
12	.000000	.357969	.348507	.000000	.000000	.000000
13	.000000	.279446	-.324509	.000000	.000000	.000000
14	.000000	.129160	-.750431	-.020603	.000000	.000000
15	.000000	-.043367	-.835617	.007233	.000000	.000000
31	.000000	-.000006	-.619701	.026518	-.002059	.000000
33	.000000	-.000004	-.275468	.026325	.001221	.000000
35	.000000	.000000	.000000	.000000	.000000	.000000

C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 3
PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

M O D E S H A P E S

MODE SHAPE NUMBER 3 PERIOD = .002905 SECONDS

DISPLACEMENTS "U" AND ROTATIONS "R"

JOINT	U(X)	U(Y)	U(Z)	R(X)	R(Y)	R(Z)
11	.000000	.658590	.032169	.000000	.000000	.000000
12	.000000	.834738	.003871	.000000	.000000	.000000
13	.000000	.763179	.011323	.000000	.000000	.000000
14	.000000	.669757	.019646	-.002431	.000000	.000000
15	.000000	.533366	.000009	.000005	.000000	.000000
31	.000000	.374235	.000001	-.000000	.000000	.001475
33	.000000	.191931	.000004	-.000000	.000000	.001198
35	.000000	.000000	.000000	.000000	.000000	.000000

(Output truncated for brevity.)

Figure VIII-8
Typical Joint Output File "EXAMPLE.SOL" Eigenvector Displacements

```
C S I / S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE    5
                                               PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

J O I N T   D I S P L A C E M E N T S

LOAD COMBINATION  1 -  DISPLACEMENTS "U" AND ROTATIONS "R"

JOINT      U(X)        U(Y)        U(Z)        R(X)        R(Y)        R(Z)
    6    .000000     .000000     .076101     .000000     .000000     .000000
    7    .000000     .000000     .059955     .000000     .000000     .000000
    8    .000000     .000000     .044475     .000000     .000000     .000000
    9    .000000     .000000     .030705    -.001106     .000000     .000000
   10    .000000     .000000     .018133    -.000911     .000000     .000000
   31    .000000     .000000     .008650    -.000666    -.000041     .000000
   33    .000000     .000000     .002309    -.000361    -.000033     .000000
   35    .000000     .000000     .000000     .000000     .000000     .000000

C S I / S A P 9 0         FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE    6
                                               PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

J O I N T   D I S P L A C E M E N T S

LOAD COMBINATION  2 -  DISPLACEMENTS "U" AND ROTATIONS "R"

JOINT      U(X)        U(Y)        U(Z)        R(X)        R(Y)        R(Z)
    6    .000000     .000031     .003087     .000000     .000000     .000000
    7    .000000     .000031     .002483     .000000     .000000     .000000
    8    .000000     .000029     .001887     .000000     .000000     .000000
    9    .000000     .000024     .001316     .000045     .000000     .000000
   10  .0000E+00   .1953E-04   .8088E-03   .3891E-04   .7870E-09   .4084E-10
   31  .0000E+00   .1371E-04   .3943E-03   .2955E-04   .1661E-05   .5415E-07
   33  .0000E+00   .7029E-05   .1143E-03   .1660E-04   .1445E-05   .4307E-07
   35    .000000     .000000     .000000     .000000     .000000     .000000

C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE    7
                                               PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

J O I N T   D I S P L A C E M E N T S

LOAD COMBINATION  3 -  DISPLACEMENTS "U" AND ROTATIONS "R"

JOINT      U(X)        U(Y)        U(Z)        R(X)        R(Y)        R(Z)
    6    .000002     .000031     .079183     .000000     .000000     .000000
    7    .000000     .000031     .062438     .000000     .000000     .000000
    8    .000000     .000029     .046363     .000000     .000000     .000000
    9    .000000     .000024     .031603    -.001061     .000000     .000000
   10    .000000     .000020     .018942    -.000872     .000000     .000000
   31    .000000     .000014     .008996    -.000637    -.000040     .000000
   33    .000000     .000007     .002503    -.000345    -.000031     .000000
   35    .000000     .000000     .000000     .000000     .000000     .000000
```

Figure VIII-8 (continued)
Typical Joint Output File "EXAMPLE.SOL"
Static Load Displacements


```
C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE    8
                                               PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

R E A C T I O N S   A N D   A P P L I E D   F O R C E S

LOAD COMBINATION  1 -  FORCES "F" AND MOMENTS "M"

JOINT      F(X)        F(Y)        F(Z)        M(X)        M(Y)        M(Z)
   35  .5104E-10   .9822E-08   .1000E+04   .8400E+05   .1929E+04   .1294E-09
   36  [illegible]  .9822E-08  -.1000E+04   .8400E+05   .1929E+04   .1519E-08

C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE    9
                                               PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

R E A C T I O N S   A N D   A P P L I E D   F O R C E S

LOAD COMBINATION  2 -  FORCES "F" AND MOMENTS "M"

JOINT      F(X)        F(Y)        F(Z)        M(X)        M(Y)        M(Z)
   35     .6769    126.5206     68.3990   3982.1442     85.7247     1.6925
   36    2.0137    126.5206     68.3990   3982.1442     85.7241    19.8.88

C S I / S A P 9 0 - - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE   10
                                               PROGRAM:SAP90/FILE:example.SOL
EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

R E A C T I O N S   A N D   A P P L I E D   F O R C E S

LOAD COMBINATION  3 -  FORCES "F" AND MOMENTS "M"

JOINT      F(X)        F(Y)        F(Z)        M(X)        M(Y)        M(Z)
   35  .6769E+00   .1265E+03  -.9316E+03   .8798E+05   .2014E+04   .1692E+01
   36  .2074E+01   .1265E+03  -.9316E+03   .8798E+05   .1864E+04   .1982E+02
```

Figure VIII-8 (continued)
Typical Joint Output File "EXAMPLE.SOL"
Static Load Reactions

B.5. File EXAMPLE.SPC

This output file contains a list of spectral values as interpolated from the response-spectrum curve for the modal time periods.

A table of modal amplitudes contains the values that are factored into each corresponding mode shape to achieve the modal displacements due to the corresponding direction of excitation.

These factors are obtained by multiplying the modal participation factors in the 1, 2 and Z directions by the spectral displacement value of the corresponding mode.

Base reaction forces in the 1, 2 and Z directions for each mode and their CQC values are output.

Finally, the CQC modal correlation matrix as developed in Reference [25] is output.

Typical response-spectrum analysis output is presented in Figure VIII-6.

The output is created by the spectrum analysis segment of the program and only exists if a response-spectrum analysis is requested.

```
C S I  /  S A P 9 0  -  -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE    1
                                                 PROGRAM SAP90/FILE:example.SPC
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 S P E C T R U M   I N P U T   D A T A

 AMPLITUDE MULTIPLIER -----"S"- 386.400
 DAMPING RATIO ------------"D"     .050
 ANGLE OF S1 WITH X-AXIS -"A"-     .0.0

C S I  /  S A P 9 0  -  -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE    2
                                                 PROGRAM SAP90/FILE:example.SPC
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

  MODE        F R E Q U E N C Y                        S P E C T R A L
 NUMBER  RAD./SEC CYCLES/SEC PERIOD-SEC (D) ACCELERATION  VELOCITY DISPLACEMENT

     1     197.02      31.36     .031892(1)          .000         000          0.0
                                        (2)       115.920         588          0.1
                                        (3)        77.666         394          0.2

     2    1178.22     187.52     .005333(1)          .000         000          0.0
                                        (2)       115.920         098          0.0
                                        (3)        77.666        .066          0.0

     3    2162.56     344.18     .002905(1)          .000        .000          0.0
                                        (2)       115.920         054          0.0
                                        (3)        77.666         036          0.0

     4    2967.23     472.25     .002118(1)           000          00          0.0
                                        (2)       115.9.0         039          0.0
                                        (3)        77.666         026          0.0

C S I  /  S A P 9 0  -  -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE    3
                                                 PROGRAM SAP90/FILE:example.SPC
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 M O D A L   A M P L I T U D E   F A C T O R S

  AT    .00 AND -90.00 DEGREES
  MODE   PERIOD   1-DIRECTION    2 DIRECTION    Z DIRECTION
    1     .032      .000000        .000000        00[illegible]
    2     .005      .000000        000000        000042
    3     .003      .000000        [illegible]037        .000000
    4     .002      .000000        [illegible]000        000004

C S I  /  S A P 9 0  -  -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE    4
                                                 PROGRAM SAP90/FILE:example.SPC
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 M O D A L   C O R R E L A T I O N   F A C T O R S

         1      2      3      4
   1  1.00    .00    .00    .00
   2   .00   1.00    .02    .01
   3   .00    .02   1.00    .09
   4   .00    .01    .09   1.00
```

Figure VIII-6

Typical Response Spectrum Output File "EXAMPLE.SPC"

FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 5
PROGRAM:[illegible]/example.SPC
[illegible] EXAMPLE 2A CANTILEVER BEAM USING EACH TYPE OF ELEMENT

MODE NUMBER	1-DIR FORCE	2-DIR FORCE	3-DIR FORCE	1-DIR MOMENT	2-DIR MOMENT	3-DIR MOMENT
1	[illegible]	[illegible]	.129298E+03	.269130E+04	.380115E-01	.384776E-01
2	[illegible]	[illegible]	.632146E+02	.289328E+04	-.894462E+00	.268886E-02
3	[illegible]	[illegible]	.256208E-03	.151636E+04	-.257576E+02	.240088E+03
4	[illegible]	.166388E-03	.149355E+02	.310752E+04	.148398E+01	-.219638E-01
[illegible]	[illegible]	.251058E+03	.136807E+01	.453118E+04	.258138E+02	.240088E+03

Figure VIII-6 (continued)
Typical Response Spectrum Output File "EXAMPLE.SPC"

B.6. File EXAMPLE.SOL

This output file contains the following:

- Mode Shapes
- Joint Displacements
- Joint Reactions and Applied Loads

The mode shapes are mass-normalized eigenvectors or Ritz vectors and exist only if an eigensolution or Ritz-vector analysis is requested.

Joint displacements, reactions (and applied loads) are for each of the static (or steady-state) load conditions and the dynamic load condition, depending upon the options activated. If load combinations are requested, the results are for the load combinations and not for the load conditions.

Due to the CQC and SRSS techniques, the results from the dynamic load condition will have no signs.

The sign conventions for joint output are defined in Figure VIII-7. Typical joint output is presented in Figure VIII-8.

This file will always exist.

```
C S I / S A P 9 0          FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE   1
                                             PROGRAM:SAP90/FILE:example.EQN
EXAMPLE   EXAMPLE 2A   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

E Q U I L I B R I U M   E Q U A T I O N   N U M B E R S
```

JOINT	X	Y	Z	XX	YY	ZZ
1	0	1	2	0	0	0
2	0	3	4	0	0	0
3	0	13	14	0	0	0
4	0	22	23	24	0	0
5	0	31	32	33	0	0
6	0	5	6	0	0	0
7	0	7	8	0	0	0
8	0	16	17	0	0	0
9	0	25	26	27	0	0
10	0	40	41	42	43	44
11	0	9	10	0	0	0
12	0	11	12	0	0	0
13	0	19	20	0	0	0
14	0	28	29	30	0	0
15	0	37	38	39	0	0
16	0	1	2	0	0	0
17	0	3	4	0	0	0
18	15	13	14	0	0	0
19	24	22	23	24	0	0
20	0	31	32	33	0	0
21	0	5	6	0	0	0
22	0	7	8	0	0	0
23	18	16	17	0	0	0
24	35	25	26	27	0	0
25	0	40	41	42	45	46
26	0	9	10	0	0	0
27	0	11	12	0	0	0
28	21	19	20	0	0	0
29	36	28	29	30	0	0
30	0	37	38	39	0	0
31	[illegible]	47	48	49	50	51
32	52	43	44	45	53	54
33	0	54	55	57	58	59
34	60	55	56	57	61	62
35	0	0	0	0	0	0
36	0	0	0	0	0	0

Figure VIII-4
Typical Equation Number File "EXAMPLE.EQN"

B.3. File EXAMPLE.EIG

This output file contains a table of the eigenvalues and corresponding frequencies and time periods.

Modal parameters related to inertial forces such as base shear and overturning moments corresponding to the global X, Y and Z directions and modal effective mass percentages (participating mass) are created.

The modal effective mass percentages corresponding to the X, Y and Z directions are given by:

$$\% \, EM_{ux} = \frac{(\Sigma M_{ux} \, \Phi_{ux})^2}{\Sigma M \, \Phi^2} * \frac{100}{\Sigma M_{ux}}$$

$$\% \, EM_{uy} = \frac{(\Sigma M_{uy} \, \Phi_{uy})^2}{\Sigma M \, \Phi^2} * \frac{100}{\Sigma M_{uy}}$$

$$\% \, EM_{uz} = \frac{(\Sigma M_{uz} \, \Phi_{uz})^2}{\Sigma M \, \Phi^2} * \frac{100}{\Sigma M_{uz}}$$

where

$$\Sigma M \, \Phi^2 = \Sigma M_{ux} \, \Phi_{ux}^2 + \Sigma M_{uy} \, \Phi_{uy}^2 + \Sigma M_{uz} \, \Phi_{uz}^2 + \Sigma M_{rx} \, \Phi_{rx}^2 + \Sigma M_{ry} \, \Phi_{ry}^2 + \Sigma M_{rz} \, \Phi_{rz}^2 = 1.0$$

and M_{ux} represents the X-translational mass values and Φ_{ux} represents the X-translational mode-shape components, and similarly for other the translational terms M_{uy}, Φ_{uy}, and M_{uz}, Φ_{uz}, and for the rotational mass and mode-shape terms M_{rx}, Φ_{rx}, M_{ry}, Φ_{ry} and M_{rz}, Φ_{rz}.

Typical output from the eigensolver is presented in Figure VIII-5.

The file is created by the SAP90 eigensolver and only exists if an eigensolution is requested.

B.4. File EXAMPLE.RIT

This output file is identical to the EXAMPLE.EIG file in format and is created by the SAP90 Ritz-vector analysis module; it only exists if Ritz-vector analysis is requested.


```
C S I  /  S A P 9 0  - -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE   2
                                              PROGRAM SAP90/FILE example.EIG
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 E I G E N   S Y S T E M   P A R A M E T E R S

  NUMBER OF EQUATIONS                  -      42
  NUMBER OF MASSES                     -      42
  NUMBER OF VALUES TO BE EVALUATED     -       4
  SIZE OF SUBSPACE                     -       8

C S I  /  S A P 9 0  - -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE   3
                                              PROGRAM SAP90/FILE example.EIG
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 E I G E N V A L U E S   A N D   F R E Q U E N C I E S

   MODE      EIGENVALUE   CIRCULAR FREQ     FREQUENCY       PERIOD
  NUMBER    (RAD/SEC)**2     (RAD/SEC)    (CYCLES/SEC)      (SEC)
     1      .3961516E+05   .1990155E+03     31.395981      .031852
     2      .1360821E+07   .1166522E+04    185.659798      .005386
     3      .4676632E+07   .2162556E+04    344.181567      .002905
     4      .8804468E+07   .2967233E+04    472.249915      .002118

C S I  /  S A P 9 0  - -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE   4
                                              PROGRAM SAP90/FILE example.EIG
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 B A S E   F O R C E   R E A C T I O N   F A C T O R S

 MODE PERIOD      X          Y          Z          X          Y          Z
   # (SEC)   DIRECTION  DIRECTION  DIRECTION    MOMENT     MOMENT     MOMENT
   1   .032 -.219E-10 -.805E-11  .329E+01   .269E+02   .780E 03   .785E 05
   2   .005  .396E-05 -.355E-05 -.744E+00 -.499E+02  -.154E 01  -.133E 03
   3   .003  .231E 03  .346E+01 -.913E-06 -.886E+01   .150E+00   .140E+03
   4   .002  .170E-06  .718E-05  .439E+00   .325E+02   .434E 01   .645E 03

C S I  /  S A P 9 0  - -  FINITE  ELEMENT  ANALYSIS  OF  STRUCTURES  PAGE   5
                                              PROGRAM SAP90/FILE example.EIG
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 P A R T I C I P A T I N G   M A S S - (percent)

 MODE      X-DIR      Y-DIR     Z-DIR       X SUM     Y SUM     Z SUM
   1        .000       .000    65.717        .000      .000    65.717
   2        .000       .000    22.081        .000      .000    87.798
   3        .092     86.511      .000        .092    86.511    87.798
   4        .000       .000     7.621        .092    86.511    95.419
```

Figure VIII-5

Typical Eigenvalue Analysis Output File "EXAMPLE.EIG"

```
C S I / S A P 9 0        FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  25
                                        PROGRAM:SAP90/FILE:example.SAP
 EXAMPLE   EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 MATERIAL ID      1

 TEMPERATURE =     CC  STRAIN STRESS RELATIONSHIP TIMES   3000000.

 { e11 }    { 1.000   .250    .250    .000   .000   .000 } { S11 }       {  .00E+00}
 { e22 }    {  .250  1.000    .250    .000   .000   .000 } { S22 }       {  .00E+00}
 { e33 } =  {  .250   .250   1.000    .000   .000   .000 } { S33 } + DT{  .00E+00}
 { e12 }    {  .000   .000    .000   2.500   .000   .000 } { S12 }       {  .00E+00}
 { e13 }    {  .000   .000    .000    .000  2.500   .000 } { S13 }       {  .00E+00}
 { e23 }    {  .000   .000    .000    .000   .000  2.500 } { S23 }       {  .00E+00}

C S I / S A P 9 0        FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  26
                                        PROGRAM:SAP90/FILE:example.SAP
 EXAMPLE   EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 S O L I D   E L E M E N T   D A T A

   ELE  JOINT  JOINT  JOINT  JOINT  JOINT  JOINT  JOINT  JOINT MAT INCMP    REF
    ID      1      2      3      4      5      6      7      8  ID MODES   TEMP

     1      3     18      8     23      4     19      9     24   1     0    .00
     2      8     23     13     28      9     24     14     29   1     0    .00

C S I / S A P 9 0        FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  27
                                        PROGRAM:SAP90/FILE:example.SAP
 EXAMPLE   EXAMPLE 24   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 T O T A L   W E I G H T S   A N D   M A S S E S

 TYPE          WEIGHT          MASS
    1           .0000          .3882

 TOTAL          .0000          .3882

C S I / S A P 9 0 - -    FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE
                                        PROGRAM:SAP90/FILE:example..
 EXAMPLE   EXAMPLE 24   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 J O I N T   L O A D S

 JOINT  LOAD         FX          FY          FZ          MX          MY          MZ
     1     1   .000E+00    .000E+00   167E+03   .000E+00   .000E+00   .000E+00
    11     1   .000E+00    .000E+00   167E+03   .000E+00   .000E+00   .000E+00
     6     1   .000E+00    .000E+00   666E+03   .000E+00   .000E+00   .000E+00
    16     1   .000E+00    .000E+00   167E+03   .000E+00   .000E+00   .000E+00
    26     1   .000E+00    .000E+00   167E+03   .000E+00   .000E+00   .000E+00
    21     1   .000E+00    .000E+00   666E+03   .000E+00   .000E+00   .000E+00
```

Figure VIII-3 (continued)
Typical Input Echo File "EXAMPLE.SAP"


```
C S I / S A P 9 0 - -  FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  29
                                      PROGRAM:SAP90/FILE:example.SAP
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 R E S P O N S E   S P E C T R U M   D A T A

 ANGLE OF S1 WITH X-AXIS            .000
 AMPLITUDE MULTIPLIER            386.400
 DAMPING RATIO                      .050
 DIRECTIONAL COMBINATION            SRSS

    PERIOD          S1          S2          S3
      .000        .000        .300        .201
      .050        .000        .300        .201
      .200        .000        .900        .603
      .300        .000        .900        .603
     1.000        .000        .300        .201
    10.000        .000        .000        .000

C S I / S A P 9 0 - -  FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  30
                                      PROGRAM:SAP90/FILE:example.SAP
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 L O A D   C O N D I T I O N   C O M B I N A T I O N   M U L T I P L I E R S

 COMBINATION   CONDITION      FACTOR
           1
                       A       1.000
           2
                       D       1.000
           3
                       1       1.000
                       D       1.000

C S I / S A P 9 0 - -  FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE  31
                                      PROGRAM:SAP90/FILE:example.SAP
 EXAMPLE - EXAMPLE 24 - CANTILEVER BEAM USING EACH TYPE OF ELEMENT

 J O I N T   A N D   E L E M E N T   O U T P U T   S E L E C T I O N

 TYPE                      FIRST     LAST      ID     CODE
                              ID       ID     INC
    1  (JOINT DISPL)           6       10       1       1
    1  (JOINT DISPL)          31       35       2       1
    2  (JOINT REACT)          35       36       1       1
    3  (JOINT MODES)          11       15       1       1
    3  (JOINT MODES)          31       35       2       1
    4  (JOINT STRESS)         13       14       1       1
    5  (FRAME ELEMT)           1        2       1       1
    6  (SHELL ELEMT)           1        4       1       1
    7  (ASOLID ELEMT)          1        1       1       1
    8  (SOLID ELEMT)           1        1       1       1
```

Figure VIII-3 (continued)
Typical Input Echo File "EXAMPLE.SAP"

```
      S A P 9 0          FINITE ELEMENT ANALYSIS OF STRUCTURES  PAGE  32
                                        PROGRAM:SAP90/FILE:example.SAP
[illegible]   EXAMPLE [illegible]   CANTILEVER BEAM USING EACH TYPE OF ELEMENT

     F I L E S   C R E A T E D   B Y   P R O G R A M

     INPUT DATA ECHO                              example.SAP
     SOLUTION ERRORS AND WARNINGS                 example.ERR
     EQUATION NUMBERING                           example.EQN
     FREQUENCIES                                  example.EIG
     SPECTRAL ANALYSIS MODAL FACTORS              example.SPC
     DISPLACEMENTS AND REACTIONS                  example.SOL
     ELEMENT FORCES                               example.F[illegible]F
     FRAME ELEMENT FORCES                         example.F3F
     SHELL ELEMENT FORCES                         example.F4F
     ASOLID ELEMENT STRESSES                      example.F5F
     SOLID ELEMENT STRESSES                       example.F8F
```

Figure VIII-3 (continued)
Typical Input Echo File "EXAMPLE.SAP"

B.2. File EXAMPLE.EQN

This output file contains a table of equation numbers assigned to the joint degrees of freedom. This table is primarily used for referring equation numbers used in warning and error messages from the equation solver back to the joint degrees of freedom.

Only active degrees of freedom are assigned equation numbers. Equation numbers run from one (1) to the total number of equations. Inactive (restrained) degrees of freedom are indicated by an equation number of zero. Active degrees of freedom that are connected by a constraint (Chapter IV, Section C) or by a rigid-floor diaphragm (Chapter IV, Section E) may share the same equation number.

The program automatically optimizes the numbering of the equations to reduce the storage required by the stiffness matrix, and to reduce the computation time required to solve the equations. For this reason, the equation numbers might not be assigned to the joint degrees of freedom in consecutive order.

Typical equation number output is presented in Figure VIII-4

This file is always created.

Variable	Note	Default	Entry
Asolid Element Location Data			
nel	(13)		Element identification number
j1, j2,...	(14)		Element joint numbers, JN option (general element definition)
j1, j3,...	(14)		Element joint numbers, JQ option (4-node element definition)
ji, jj,...	(14)		Element joint numbers, JS option (regular mesh definition)
mat	(15)	[pv]	Element material type
tz	(16)	[0]	Zero-stress reference temperature
th	(17)	[pv]	Element thickness
g1, g2	(18)		Element generation parameters
n	(19)	[2]	Element plane identifier =1 X-Y plane =2 Y-Z plane =3 Z-X plane

NOTES

1. The control parameter **nmat** defines the number of data lines the program expects to read in the material property data section (Format Section c).

2. The entry et defines the type of element that is to be formulated. The element type defined here applies to all elements in the ASOLID data block. See Figure X-20.

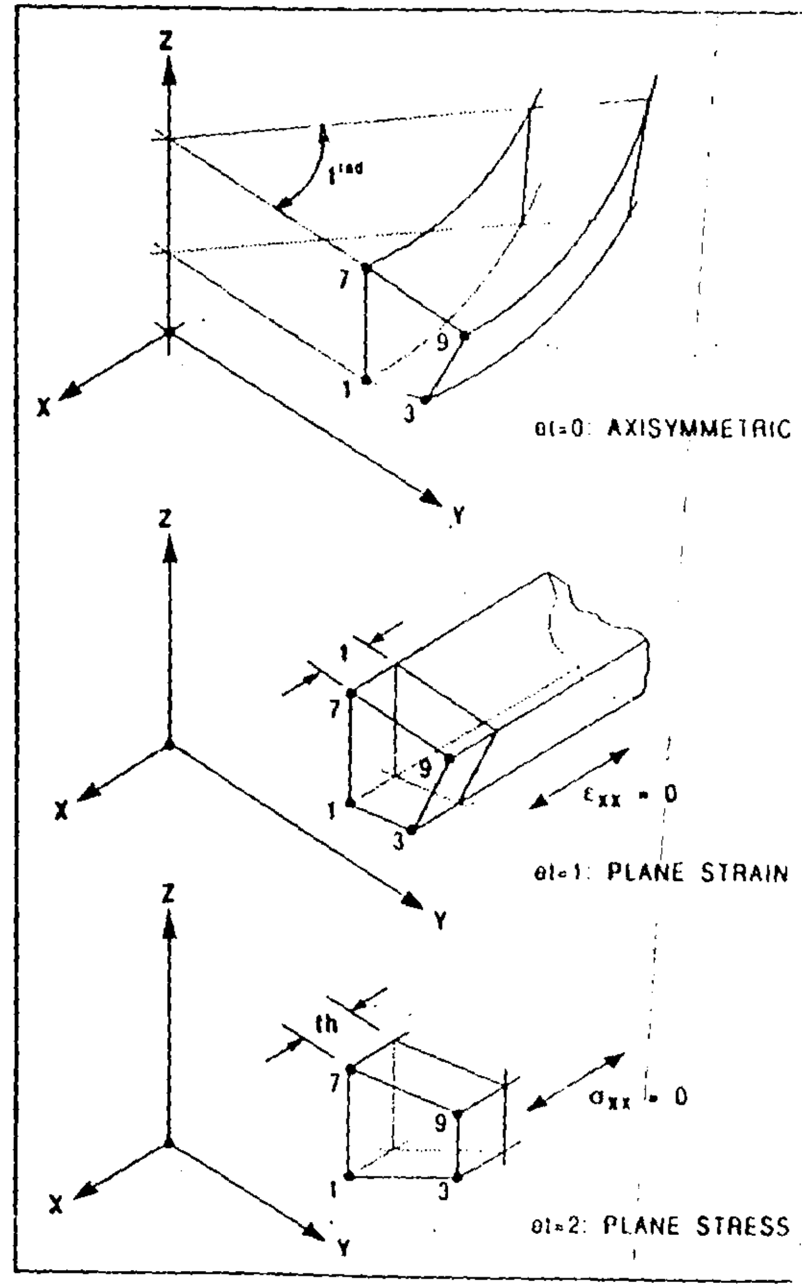


Figure X-20
ASOLID Element Types

3. The entry **ntm** is required in order to pre-allocate memory storage for the temperature-dependent material properties defined in the material property data.

4. The **nld** load multipliers associated with the X-direction ($x_1, x_2, ..., x_{nld}$) correspond to the **nld** structural load conditions. These are gravitational multipliers that activate the self weight of the ASOLID elements in the X-direction. In other words, static loads acting in the X-direction equal to the self-weight of the ASOLID elements, factored by the gravitational multipliers, will be added to the corresponding load conditions.

   For example, if $x_2 = 1.4$, static loads consisting of 1.4 times the weight of the ASOLID elements, acting in the positive global X-direction, will be added to the structural load condition 2. Note that only elements that have non-zero weights per unit volume contribute to the static vectors. Similarly, the **nld** Y-direction and Z-direction multipliers generate self-weight vectors in the Y-direction and Z-direction, respectively.

   Because all structures defined by this element are two-dimensional, gravitational multipliers in a direction normal to the element will have no effect.

   The **nld** temperature multipliers ($t_1, t_2, ..., t_{nld}$) and pressure multipliers ($p_1, p_2, ..., p_{nld}$) similarly generate thermal and pressure body forces for the corresponding load conditions using the joint temperature and pressure values specified in the POTENTIAL data block.

5. Values for angular velocity, in radians per unit time, may only be specified for axisymmetric elements. The whole structure is assumed to rotate with one angular velocity for

a particular load condition. The angular rotation is assumed to be about the global Y-axis, Z-axis, and X-axis for elements in the X-Y, Y-Z and Z-X planes, respectively. Radial loads are calculated using calculated radial accelerations (from the angular velocities) and the structural masses.

6. The material property identification numbers must be in ascending consecutive numerical sequence starting with the number one (1).

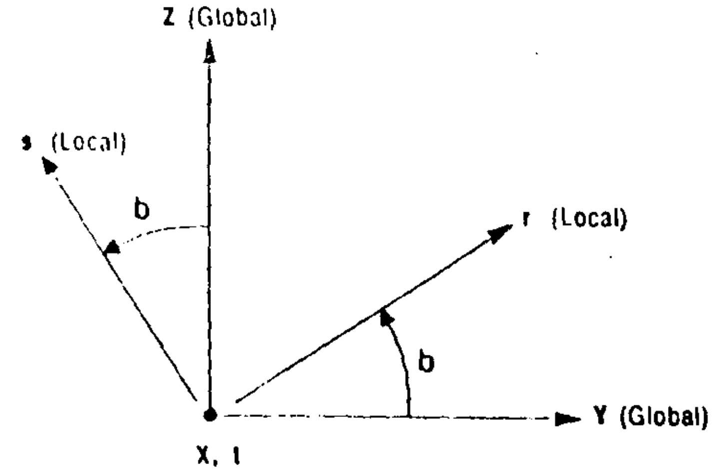
7. This entry defines the number of data lines the program expects to read in Format Section c(ii). The value of **nt** must not be greater than **ntm**, described in Format Section b.

8. The weight per unit volume is used for calculating the self-weight of the element. The self-weight is added into the structural load conditions via the gravitational load multipliers described above in Note 4.

9. The mass per unit volume is used for the calculation of the mass of the element. Consistent mass units must be used. This entry is needed for automatic lumping of the element mass to the element joints when assembling the structural mass matrix. These nodal masses are needed if a dynamic analysis mode is activated, or if angular velocities are specified for axisymmetric elements. See Note 5 above.

10. This angle, in degrees, as shown in Figure X-21, defines the axes along which the orthotropic material properties are specified.

    For elements in the Y-Z plane, the local r-s directions are defined with respect to the global Y-Z system using the angle **b**, and the t-axis is the same as the X-axis.



MATERIAL PROPERTY REFERENCE SYSTEM

$$\begin{bmatrix} \epsilon_{rr} \\ \epsilon_{ss} \\ \epsilon_{tt} \\ \epsilon_{rs} \end{bmatrix} = \begin{bmatrix} \frac{1}{e_r} & \frac{-u_1}{e_s} & \frac{-u_2}{e_t} & 0 \\ \frac{-u_1}{e_s} & \frac{1}{e_s} & \frac{-u_3}{e_t} & 0 \\ \frac{-u_2}{e_t} & \frac{-u_3}{e_t} & \frac{1}{e_t} & 0 \\ 0 & 0 & 0 & \frac{1}{g_{rs}} \end{bmatrix} \begin{bmatrix} \sigma_{rr} \\ \sigma_{ss} \\ \sigma_{tt} \\ \sigma_{rs} \end{bmatrix} + \begin{bmatrix} a_r\,dT \\ a_s\,dT \\ a_t\,dT \\ 0 \end{bmatrix}$$

MATERIAL PROPERTY LAW

Figure X-21
ASOLID Element Material Properties

Similarly, for elements in the X-Y and Z-X planes, the local r-s directions are defined with respect to the global X-Y and Z-X planes. The t-axis is the same as the Z- and Y-axis, respectively.

11. The values of **t** on consecutive data lines within any one material property data set, must be in numerically ascending order. If joint temperatures are defined via the POTENTIAL data block, the range of these **t** values must encompass all the temperature specifications that exist in the POTENTIAL data block.

12. These material properties correspond to the temperature **t** specified on this data line.

    If values for **e**, **u** or **a** are not specified in the s- and t-directions, they will be set equal to the values specified for the r-direction. Therefore, for isotropic materials, only $\mathbf{e_r}$, **u1** and $\mathbf{a_r}$ need to be input.

13. The element identification number can be any number between 1 and **nid** (SYSTEM data block). Element numbers do not have to be consecutive and may be supplied in any order.

    Elements may be re-specified or re-generated, in which case only the last definition is used. When an element is redefined the previous definition is completely lost; all unspecified variables use the standard default values, and "previous-value" defaults refer to the previous data line, not to the previous definition of the element being redefined.

    A previously defined element can be deleted by setting **jid** to the *negative* of its identification number. This may be used, for example, to create gaps within regions of generated elements. The only other data permitted on the data line when deleting elements is G=g_1 which specifies the total number of elements to be deleted; the element identification numbers increment by 1. See Note 18.

14. In general, the ASOLID element is defined by specifying the nine joints **$j_1 j_2 j_3 j_4 ... j_9$** with the JN identifier. The input sequence of the joint numbers that describe the elements should be as shown in Figure X-22. Any element joints that do not exist should be given values of zero. Therefore, the four-node quadrilateral may be specified as follows:

JN=j_1,0,j_3,0,0,0,j_7,0,j_9

However, a special input option for defining four-node elements is available by using the the JQ identifier in place of the JN option. By this method a four-node quadrilateral is specified as follows:

JQ=$j_1 j_3 j_7 j_9$

Three-node triangular elements cannot be specified using the JN options but should be specified as follows:

JQ=$j_1 j_3 j_7$

Simple nine-node elements which have constant joint increments in both directions can be defined by using the JS identifier. By this method a nine-node element can be defined as follows:

JS=$j_i j_j j_k$

where $j_i = j_1$, $j_j = j_2$, and $j_k = j_4$

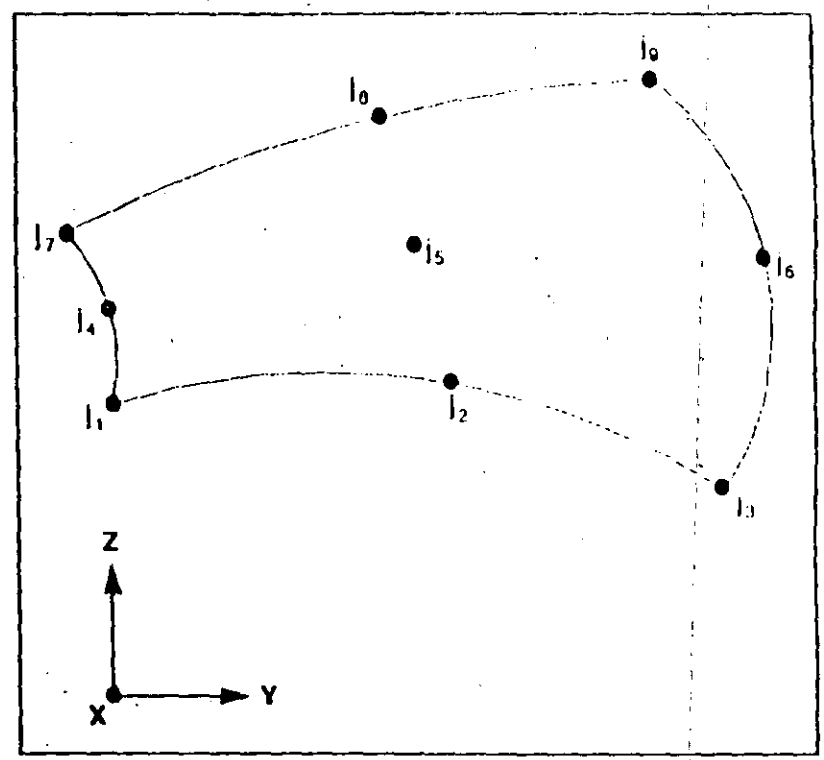


Figure X-22
Variable Three- to Nine-Node
Two-dimensional Isoparametric Element

The joint numbers **$j_3 j_5 j_6 j_7 j_8 j_9$** are assumed to be as follows:

$j_3 = j_2 + I$
$j_5 = j_4 + I$
$j_6 = j_5 + I$
$j_7 = j_4 + J$
$j_8 = j_7 + I$
$j_9 = j_8 + I$

where $I = j_2 - j_1$ and $J = j_4 - j_1$.

Only one of the JN, JQ or JS identifiers may exist on any one data line. All joint numbers should have been defined in the JOINTS data block.

15. **mat** refers back to the material table defined in Format Section c; **mat** must be a positive number not greater than **nmat**.

If joint temperatures are specified via the POTENTIAL data block, an element material temperature value equal to the average of the element joint temperatures is calculated.

The properties assigned to the element are then computed by linearly interpolating the properties associated with material type **mat** to obtain the properties at the element material temperature.

16. The joint temperatures specified in the POTENTIAL data block are used to induce thermal load in the elements. The element zero-stress reference temperature is subtracted from the element joint temperatures defined in the POTENTIAL data block to compute the temperature differences that produce the thermal strains. Therefore, if a structure is heated to 600 degrees and the zero-stress reference temperature is 100 degrees, the thermal strains will be based upon a temperature increase of 500 degrees.

17. The element thickness is only required for plane-stress elements. A unit value is used for axisymmetric and plane-strain elements as shown in Figure X-20.

The element thickness is used for calculating the element stiffness as well as the element volume for the element self-weight and mass calculation.

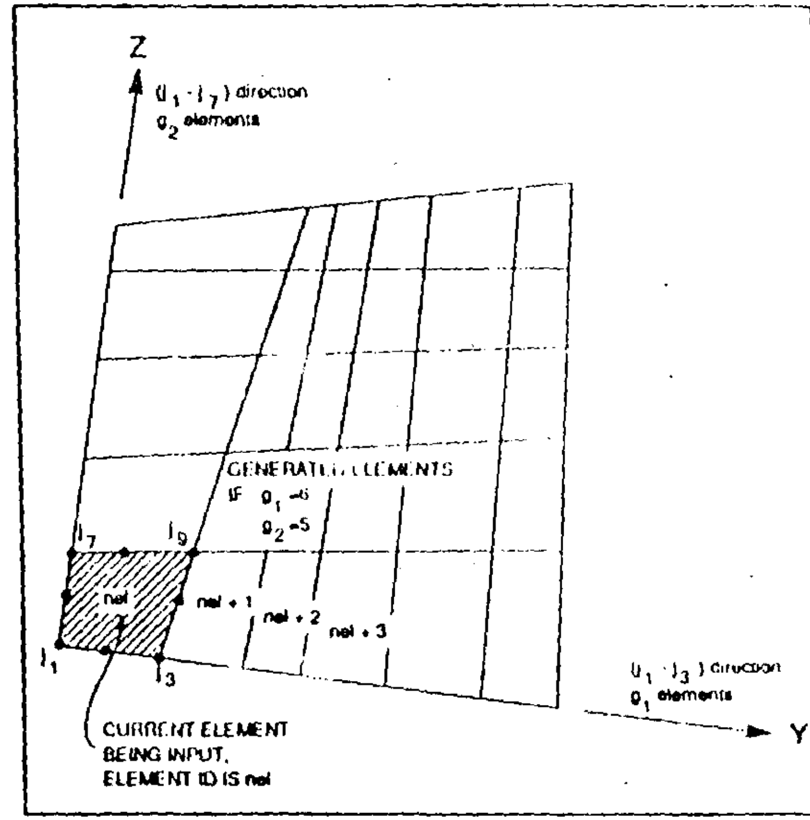


Figure X-23
ASOLID Element Generation

18. **g1** and **g2** are parameters that cause the generation of a two-dimensional mesh with **g1** elements in the (j1–j3) direction and **g2** elements in the (j1–j7) direction. The value of **g1** must not be less than 1, since this number includes the current element being defined; similarly for **g2**. See Figure X-23.

The node numbers of the generated elements are formed by incrementing the node numbers of the basic element by (j3–j1) in the j1-j3 direction and by (j7–j1) in the j1-j7 direction. Therefore, generation is restricted to meshes with regular numbering systems. The element identification numbers for the generated elements are obtained by incrementing the identification number of the previous element by 1. All the generated elements have the same material identification, element type, thickness and reference temperature.

When deleting elements only g_1 may be specified, giving the *total* number of elements being deleted. The identification numbers of the deleted elements are assumed to increment by 1. See Note 13.

19. The ASOLID element can only exist parallel to global principal planes. This entry identifies the plane to which this element is parallel. In the case of axisymmetric elements the elements must exist *in* the principal planes. The axisymmetric model is generated about the Y-axis, Z-axis and X-axis for elements in the X-Y, Y-Z and Z-X planes, respectively.

12. "SOLID" Data Block

This data block defines the properties, locations and loadings associated with the three-dimensional, eight-node SOLID element. Skip this data block if there are no SOLID elements in the model. Otherwise prepare data for Format Sections a through d as described below.

FORMAT

a. Separator

Provide one data line for the SOLID separator in the following form:

SOLID

b. SOLID Control Information

Provide one data line for the SOLID control information in the following form:

NM=nmat MAX tm
X=x_1, x_2,..., x_{nld} Y=y_1, y_2,..., y_{nld} Z=z_1, z_2,..., z_{nld}
T=t_1, t_2,..., t_{nld} P=p_1, p_2,..., p_{nld}

c. **Material Property Data**

Provide one set of data for each of the **nmat** material property types. Each data set consists of a first data line, immediately followed by temperature-dependent material property data as shown below:

(i) **First Data Line**

Prepare one data line in the following form:

nm NUMT=nt W=w M=m

(ii) **Temperature-Dependent Material Property Data**

Prepare **nt** data lines in the following form:

T=t E=e_x, e_y, e_z U=u_1, u_2, ..., u_{15} G=g_{xy}, g_{yz}, g_{zx} A=a_1, a_2, ..., a_6

d. **SOLID Element Location Data**

In this data section provide as many data lines as needed to define all the SOLID elements in the model. **End this data section with a blank line.** Prepare the data lines in the following form:

nel JQ=j_1, j_2, j_3, j_4, j_5,..., j_8 JR=j_i, j_j, j_k, j_l M=mat TZ=tz I=i G=g_1, g_2, g_3

DESCRIPTION

Variable	Note	Default	Entry
Solid Control Information			
nmat	(1)	[1]	Number of material types
ntm	(2)	[1]	Maximum number of temperatures in any one of the **nmat** material property sets
x_1,x_2...	(3)	[0]	X-direction gravitational multipliers for the **nld** structural load conditions
y_1,y_2...	(3)	[0]	Y-direction gravitational multipliers for the **nld** structural load conditions
z_1,z_2...	(3)	[0]	Z-direction gravitational multipliers for the **nld** structural load conditions
t_1,t_2...	(3)	[0]	Temperature multipliers for the **nld** structural load conditions
p_1,p_2...	(3)	[0]	Pressure multipliers for the **nld** structural load conditions

Variable	Note	Default	Entry
Material Property Data			
nm	(4)		Material identification number
nt	(5)	[1]	Number of temperatures for which material-dependent properties are specified in this material type
w	(6)	[0]	Weight per unit volume
m	(6)	[0]	Mass per unit volume
t	(7)	[0]	Temperature associated with material properties specified on this data line
e_x, e_y, e_z	(8)	[0]	Modulus of elasticity, in the x-, y- and z-directions, respectively
$u_1, u_2, ...$	(8)	[0]	Poisson's ratios
$g_{xy}, g_{yz}, ...$	(8)	[0]	Shear modulus in the X-Y, Y-Z and Z-X planes respectively
$a_1, a_2, ..$	(8)	[0]	Coefficients of thermal expansion, (L/L/H units)

Variable	Note	Default	Entry
Solid Element Location Data			
nel	(9)		Element identification number
$j_1, j_2...$	(10)		Element joint numbers, JQ option (general element definition)
$j_i, j_j...$	(10)		Element joint numbers, JR option (regular element definition)
mat	(11)	[pv]	Element material type identification number
tz	(12)	[0]	Zero-stress reference temperature
I	(13)	[pv]	Flag for suppressing incompatible bending modes
g_1, g_2, g_3	(14)		Element generation parameters

NOTES

1. The control parameter **nmat** defines the number of data sets the program expects to read in the material property data (Format Section c.
2. The entry **ntm** is required in order to pre-allocate memory storage for the temperature-depende... material properties defined in the material property data.
3. The **nld** load multipliers associated with the X-direction **($x_1, x_2, ..., x_{nld}$)** correspond to the nld structural load conditions. These are gravitational multipliers that activate the self-weight of the SOLID elements in the X-direction. In other words, static loads acting in the X-direction equal to

the self-weight of the SOLID elements, factored by the gravitational multipliers, will be added to the corresponding load condition.

For example, if x 1.4, static loads consisting of 1.4 times the weight of the SOLID elements, acting in the positive global X-direction, will be added to the structural load condition 2. Note that only elements that have non-zero weights per unit volume contribute to the static vectors. Similarly, the **nld** Y-direction multipliers generate self-weight vectors in the Y-direction and the **nld** Z-direction multipliers generate self-weight vectors in the Z-direction.

Similarly, the **nld** temperature multipliers (**$t_1, t_2, ..., t_{nld}$**) and pressure multipliers (**$p_1, p_2, ..., p_{nld}$**) generate thermal and pressure body forces for the corresponding load conditions using the nodal temperature and pressure values specified in the POTENTIAL data block.

4. The material property identification numbers must be in ascending consecutive numerical sequence starting with the number one (1).

5. This entry defines the number of data lines the program expects to read in Format Section c(ii). The value of **nt** must not be greater than **ntm**, described in Format Section b.

6. The weight per unit volume is used for calculating the self-weight of the element. The self-weight is added into the structural load conditions via the gravitational load multipliers described above in Note 3.

   The mass per unit volume is used for the calculation of the mass of the element. Consistent mass units must be used. This entry is only needed in dynamic analysis mode for automatic lumping of the element mass to the element joints when assembling the structural mass matrix.

7. The values of **t**, on consecutive data lines within any one material property data set, must be in numerically ascending order. If nodal temperatures are defined via the POTENTIAL data block, the range of these **t** values must encompass all the temperature specifications that exist in the POTENTIAL data block.

8. These material properties correspond to the temperature **t** specified on this data line.

   The material properties are defined as follows:

$$
\begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \epsilon_{xy} \\ \epsilon_{xz} \\ \epsilon_{yz} \end{bmatrix}
=
\begin{bmatrix}
\frac{1}{e_x} & \frac{-u_1}{e_y} & \frac{-u_2}{e_z} & \frac{-u_4}{g_{xy}} & \frac{-u_7}{g_{xz}} & \frac{-u_{11}}{g_{yz}} \\
 & \frac{1}{e_y} & \frac{-u_3}{e_z} & \frac{-u_5}{g_{xy}} & \frac{-u_8}{g_{xz}} & \frac{-u_{12}}{g_{yz}} \\
 & & \frac{1}{e_z} & \frac{-u_6}{g_{xy}} & \frac{-u_9}{g_{xz}} & \frac{-u_{13}}{g_{yz}} \\
 & & & \frac{1}{g_{xy}} & \frac{-u_{10}}{g_{xz}} & \frac{-u_{14}}{g_{yz}} \\
 & \text{Symmetric} & & & \frac{1}{g_{xz}} & \frac{-u_{15}}{g_{yz}} \\
 & & & & & \frac{1}{g_{yz}}
\end{bmatrix}
\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{bmatrix}
+ \Delta T
\begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \end{bmatrix}
$$

   For isotropic materials only **e_x, u_1** and **a_1** need be specified. All other values are derived from them.

9. The element identification number can be any number between 1 and **nid** (SYSTEM data block). Elements numbers do not have to be consecutive and may be supplied in any order.

Elements may be re-specified or re-generated, in which case only the last definition is used. When an element is redefined the previous definition is completely lost; all unspecified variables use the standard default values, and "previous-value" defaults refer to the previous data line, not to the previous definition of the element being redefined.

A previously defined element can be deleted by setting **jid** to the *negative* of its identification number. This may be used, for example, to create gaps within regions of generated elements. The only other data permitted on the data line when deleting elements is G=g_1 which specifies the total number of elements to be deleted; the element identification numbers increment by 1. See Note 14.

10. In general, the solid element is defined by specifying the eight joints **j_1,j_2,j_3,j_4,..., j_8** with the JQ identifier. The input sequence of the joint numbers to describe the element should be as shown in Figure X-24.

However, if the mesh is regular such that the joint increments are constant in each of the three directions, the element may be specified via the JR identifier with only four entries as follows:

$$\text{JR} = \mathbf{j_i j_j j_k j_l}$$

where $j_i = j_1$, $j_j = j_2$, $j_k = j_3$, and $j_l = j_5$

The joint numbers **j_4,j_6,j_7,j_8** are assumed to be as follows:

$$j_4 = j_2 + I$$
$$j_6 = j_2 + J$$
$$j_7 = j_3 + J$$
$$j_8 = j_4 + J$$

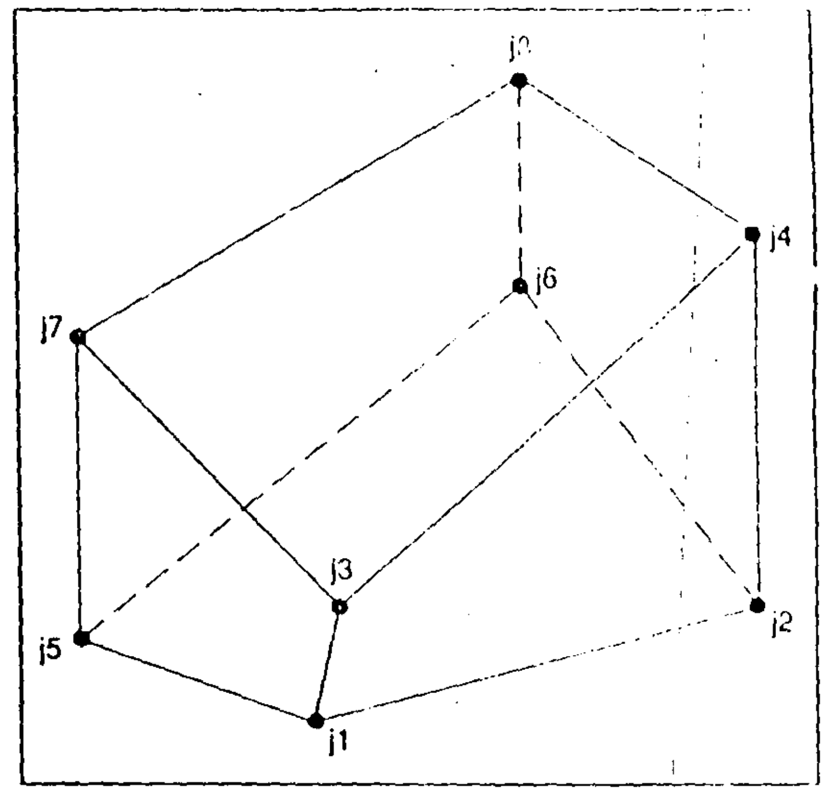


Figure X-24
Three-dimensional Solid Element

where $\mathbf{I} = \mathbf{j_3} - \mathbf{j_1}$ and $\mathbf{J} = \mathbf{j_5} - \mathbf{j_1}$.

The JQ and JR identifiers may not exist concurrently on the same data line. All joint numbers should have been defined in the JOINTS data block.

11. **mat** refers back to the material property table defined in Format Section c; **mat** must be a positive number not greater than **nmat**.

12. The joint temperatures specified in the POTENTIAL data block are used to induce thermal load in the elements. The

element zero-stress reference temperature is subtracted from the element joint temperatures defined in the POTENTIAL data block to compute the temperature differences that produce the thermal strains. Therefore, if a structure is heated to 600 degrees and the zero-stress reference temperature is 100 degrees, the thermal strains will be based upon a temperature increase of 500 degrees.

13. If **i** is equal to 0, incompatible bending modes will be included in the element formulation. These modes considerably improve the bending behavior of the element if the element geometry is of a rectangular form. If the element is severely distorted, the inclusion of the incompatible modes should be suppressed by specifying a value of **i** equal to 1.

14. g1, g2 and g3 are parameters that cause the generation of elements, forming a three-dimensional mesh with **g1** elements in the **j1-j2** direction, **g2** elements in the **j1-j5** direction and **g3** elements in the **j1-j3** direction. The value of **g1** must not be less than 1, since this number includes the current element being defined; similarly for **g2** and **g3**. See Figure X-25.

The joint numbers of the generated elements are formed by incrementing the joint numbers of the basic element by (j2-j1) in the **j1-j2** direction, by (j5-j1) in the **j1-j5** direction, and by (j3-j1) in the **j1-j3** direction. Therefore, generation is restricted to meshes with regular numbering systems. The element identification numbers for the generated elements are obtained by incrementing the identification number of the previous element by 1. All of the generated elements have the same material properties and reference temperatures.

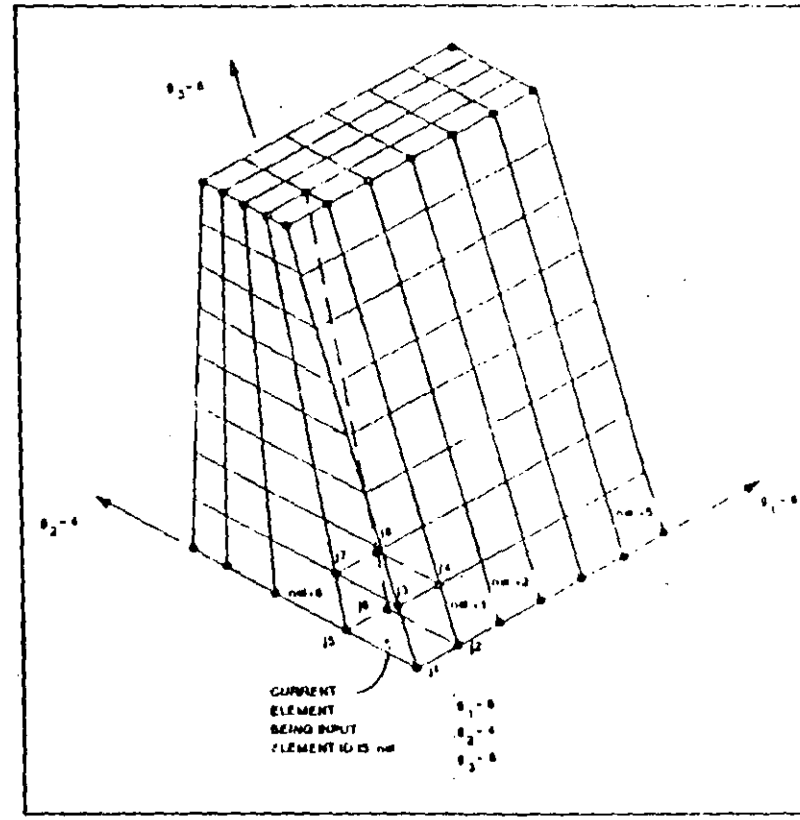


Figure X-25
Solid Element Generation

When deleting elements only **g1** may be specified, giving the *total* number of elements being deleted. The identification numbers of the deleted elements are assumed to increment by 1. See Note 9.

13. "LOADS" Data Block

Loads in the form of point forces or moments may be applied to any of the joints of the structure. This data block defines the joint loading specifications for the **nld** load conditions. Specified loads may not be applied along restrained degrees of freedom.

Skip this data block if there are no loads to be applied to the joints of the model. Otherwise prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the LOADS separator in the following form:

LOADS

b. Load Data

In this data section provide as many data lines as needed to define the joint loading for the **nld** structural load conditions. **End this data section with a blank line.** Prepare the data lines in the following form:

j1 j2 inc L=l F=f_x, f_y, f_z, m_x, m_y, m_z

EXAMPLE

LOADS
12 18 2 L=2 F=10,0

DESCRIPTION

Variable	Note	Default	Entry
j1	(1)		First joint number
j2		[j1]	Last joint number
inc		[1]	Joint number increment
l		[pv]	Load condition number
f_x	(2)		Applied force in the global X-direction
f_y			Applied force in the global Y-direction
f_z			Applied force in the global Z-direction
m_x			Applied moment about the global X-axis
m_y			Applied moment about the global Y-axis
m_z			Applied moment about the global Z-axis

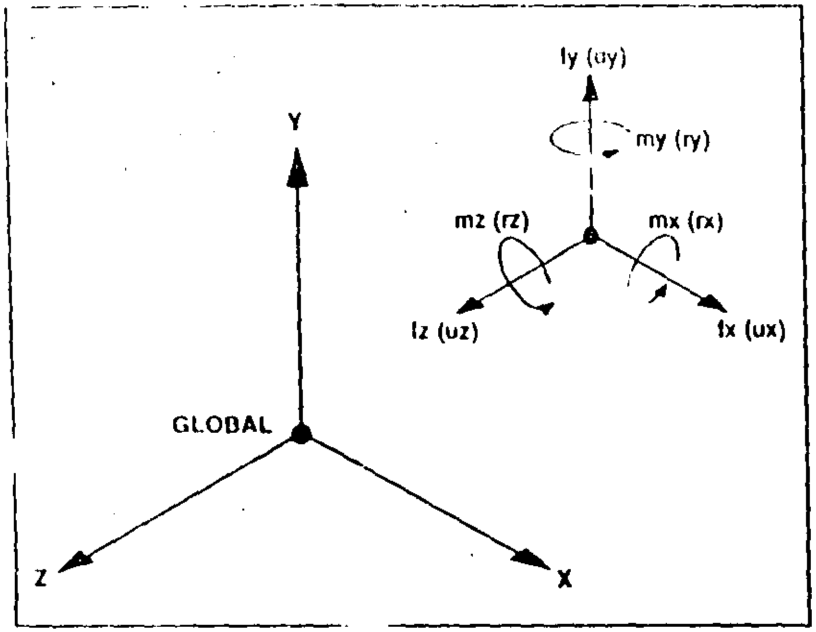


Figure X-26
Positive Sign Convention for Applied Joint Loads or Displacements

NOTES

1. The parameters j_1, j_2 and **inc** define the following series of joint numbers:

   j_1, j_1+inc, j_1+2inc, j_1+3inc,...

   which continues until j_2 is reached. All the joints in this series will receive the joint loading specified on this line for the load condition l.

2. Loads may be input in any order of the joints or the load conditions. Repeated loading specifications for a particular joint (in a specific load condition) are algebraically addi- See Figure X-26 for the sign convention.

14. "DISPLACEMENTS" Data Block

It is possible to apply specific translational or rotational static displacements to any of the joints of t structure.

Specified displacements may not be applied along restrained degrees of freedom.

This data block is not allowed in the dynamic analysis mode.

The displacement pattern defined by this data block will be applied to each of t... **nld** static load conditions.

It is possible to include load specifications and displacement specifications in the same run. However, non-zero displacements may not be specified along degrees of freedom that have non-zero load specifications.

Skip this data block if there are no displacements to be applied to the joints of the model. Otherwise prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the DISPLACEMENTS separator in the following form:

DISPLACEMENTS

b. Displacement Data

In this data section provide as many data lines as needed to define the applied joint displacements. **End this data section with a blank line.** Prepare the data lines in the following form:

j_1 j_2 inc U=u_x, u_y, [illegible] r_y, r_z

EXAMPLE

DISPLACEMENTS
12 18 2 U=1,0,0

DESCRIPTION

Variable	Note	Default	Entry
j_1	(1)		First joint number
j_2		[j_1]	Last joint number
inc		[1]	Joint number increment
u_x	(2)		Applied displacement in the global X-direction
u_y			Applied displacement in the global Y-direction
u_z			Applied displacement in the global Z-direction
r_x			Applied rotation about the global X-axis
r_y			Applied rotation about the global Y-axis
r_z			Applied rotation about the global Z-axis

NOTES

1. The parameters **j1**, **j2** and **inc** define the following series of joint numbers:

   j1, j1+inc, j1+2inc, j1+3inc,...

   which continues until **j2** is reached. The same displacements will be applied to all the joints in this series for all of the **nld** load conditions.

2. Displacements may be input in any order of the joints. Repeated specifications for a particular joint are algebraically additive. See Figure X-26 for the sign convention.

   Only the degrees of freedom with nonzero displacement specifications will be processed. In other words, a displacement value of zero may not be specified for a degree of freedom using the DISPLACEMENTS data block. If a zero displacement is required, the user must restrain the degree of freedom using the RESTRAINTS data block.

15. "PRESTRESS" Data Block

Any of the FRAME elements (beams) in the model can be subjected to loading patterns produced by prestressing cables.

This data block is for defining such loads in a convenient and statically consistent manner.

The addition of these loads into the basic load conditions is achieved via the prestress multipliers in the FRAME data block, on the FRAME control data line.

Skip this data block if there are no prestress loads to be applied to any of the beams in the model. Otherwise, prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the PRESTRESS separator in the following form:

PRESTRESS

b. Prestress Data

In this data section provide as many data lines as needed to define the prestress cable geometry and prestress forces for the beams. **End this data section with a blank line.** Prepare the data in the following form:

nb1 nb2 ninc D=d_i, d_c, d_j T=t

EXAMPLE

```
PRESTRESS
1 12 1   D=0.25,0.50,0.5   T=100
```

Remember that consistent units are required.

DESCRIPTION

Variable	Note	Default	Entry
nb_1	(1)		First FRAME element number
nb_2		$[nb_1]$	Last FRAME element number
nine		[1]	FRAME element number increment
d_I	(2)		Cable drape at End I
d_c			Cable drape at center
d_J			Cable drape at End J
t			Prestress tension force

NOTES

1. The parameters **nb₁**, **nb₂** and **ninc** define the following series of FRAME element numbers:

   nb_1, nb_1+ninc, nb_1+2ninc, nb_1+3ninc,...

   which continues until **nb_2** is reached.

   All the FRAME elements in this series will receive the cable drape and prestressing tension described by this line.

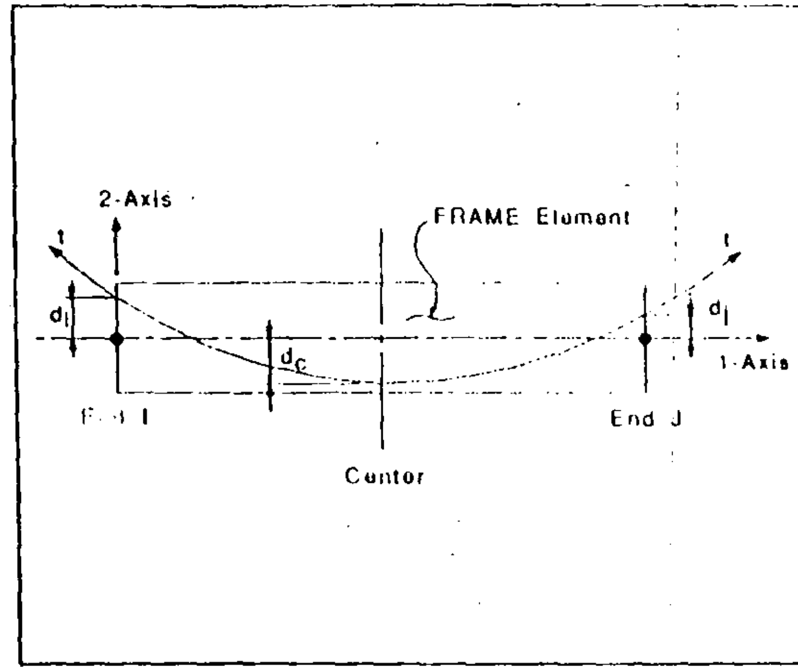


Figure X-27
ress Cable Profile

   FRAME element duplications are additive. Therefore, if more than one cable exists in the same element the above information must be repeated for each cable. The FRAME element numbers may be input in any order.

2. These dimensions are with respect to the neutral axis in the element local 1-2 plane. A positive drape is measured in the positive 2 direction, as shown in Figure X-27.

   The drape configuration is assumed to be parabolic. The cable tension is assumed to be constant throughout.

16. "PDELTA" Data Block

The PDELTA data block causes the transverse bending stiffness of all FRAME elements to be modified to account for the P-Delta effect under large axial force.

An iterative P-Delta analysis is performed to determine the effective stiffness under a single combination of static loads as defined in the PDELTA data block. This effective stiffness is then used for all other analyses in the run, including static load conditions, combinations, envelopes, eigen and Ritz vectors, response-spectrum and time-history analyses, bridge influence lines and bridge moving-load cases. It should not generally be used with steady-state analysis.

See Chapter VI for a complete description of the P-Delta analysis option. Some of the more important assumptions are summarized here:

- The P-Delta effect is analyzed only in FRAME elements. Other types of elements may still be present in the model.
- Only the large-stress effect of an axial force upon transverse bending and shear deformation is considered.
- All deflections, strains and rotations are assumed to be small.
- The transverse deflected shape of a FRAME element is assumed to be cubic in bending and linear in shear between the reduced rigid zone offsets.
- P-Delta axial forces are assumed to be constant along element length.

- Axial prestress forces from the PRESTRESS data block do not contribute to the P-Delta effect.

Note that including a P-Delta analysis may considerably increase computation time and may make interpretation of the results more difficult. It is *strongly recommended* that a preliminary linear analysis be performed to check the model for correctness before using the PDELTA data block.

Skip this data block if a P-Delta analysis is not to be performed. Otherwise, prepare data for Format Sections a, b and c as described below.

FORMAT

a. Separator

Provide one data line for the PDELTA separator in the following form:

PDELTA

b. Control Information

Prepare one data line in the following form:

M=m TOLD=told TOLP=tolp

c. P-Delta Load Combination Factors

In this data section, provide up to **nld** data lines to define the P-Delta load combination. **End this data section with a blank line.** Prepare each data line in the following form:

L=l SF=**sf**

EXAMPLE

Suppose that load conditions 1 and 3 are dead load and live load, respectively. The following data specifies the P-Delta load combination to be 1.2 times the dead load plus 0.5 times the live load. No other load conditions are included.

```
PDELTA
M=5   TOLD=0.0001
L=1   SF=1.2
L=3   SF=0.5
```


DESCRIPTION

Variable	Note	Default	Entry
Control Information			
m	(1)	[1]	Maximum number of additional iterations
told	(1)	[.001]	Relative displacement convergence tolerance
tolp	(1)	[.001]	Relative force convergence tolerance *Currently not active - ignored*
P-Delta Load Combinations Factors			
l	(2)		Load condition number
sf	(2)	[0]	Multiplier for static load condition **l**

NOTES

1. The program always performs an intial, or zero-th, iteration which is a standard linear, static analysis. Up to **m** additional iterations will be performed correcting the stiffness matrix for the P-Delta effect under the P-Delta load combination. Iteration will stop when **m** additional iterations have been performed, or when the relative change in displacement from one iteration to the next is less than the tolerance **told**, whichever occurs first.

   Setting **m**=0 effectively turns the P-Delta analysis off, and is the same as omitting the PDELTA data block. Setting **told**=0 will force all **m** additional iterations to be performed.

The relative force convergence tolerance **tolp** is reserved for future use and is not currently used.

For more detail, see the description of the Iterative Analysis Procedure in Chapter VI, Section C.1.

2. The P-Delta load combination is defined as the sum of the specified static load conditions, each multiplied by the specified scale factor **sf**. Any of the **nld** basic load conditions that are omitted are not added into the P-Delta load combination.

   Each load condition should be specified at most once. If a load condition is repeatedly specified, the scale factor from the last specification is used.

   For more detail, see the description of the P-Delta Load Combination in Chapter VI, Section C.1.

17. "SPEC" Data Block

This data block is for defining the data associated with the response-spectrum dynamic analysis. The results of this analysis define the **dynamic load condition**.

Skip this data block if a response-spectrum dynamic analysis of the structure is not desired. Otherwise, prepare data for Format Sections a, b and c as described below. This data block requires that **nfq** or **nritz** be specified in the SYSTEM data block.

FORMAT

a. Separator

Provide one data line for the SPEC separator in the following form:

SPEC

b. Control Information

Prepare one data line in the following form:

A=a S=s D=**d**

c. Spectrum Curve Data

In this data section, provide as many data lines as needed to define the response-spectrum curve. **End this data section with a blank line.** Prepare each data line in the following form:

tp s1 s2 sz

EXAMPLE

```
SPEC
A=45   S=386.4   D=.05
  .0    .30    .30    .30 * .67
  .1    .35    .35    .35 * .67
  .2    .70    .70    .70 * .67
  .5    .90    .90    .90 * .67
  .6    .90    .90    .90 * .67
 1.0    .60    .60    .60 * .67
 2.0    .50    .50    .50 * .67
100.    .00    .00    .00
```


DESCRIPTION

Variable	Note	Default	Entry
Control Information			
a	(1)	[0.0]	excitation angle, degrees
s	(2)	[1.0]	Response-spectrum scale factor
d	(3)	[0.0]	Structural damping ratio
Spectrum Curve Data			
tp	(4)		Time period
s1	(4)		Spectrum value in 1-direction
s2	(4)	[0]	Spectrum value in 2-direction
sz	(4)	[0]	Spectrum value in Z-direction

NOTES

1. The ground acceleration may be applied simultaneously in three directions, namely, two mutually perpendicular directions in the X-Y plane and the Z-direction. The two directions in the X-Y plane are defined as 1 and 2 where the 1-direction is defined by the angle **a**, measured counterclockwise from the global X-axis. The 2-direction is defined to be normal to the 1-direction. See Figure X-28.

   Even though spectral accelerations can be specified in three directions, only one result is output for each response quantity. The results are the SRSS (square root of the sum of the squares) of the three results for the three directions.

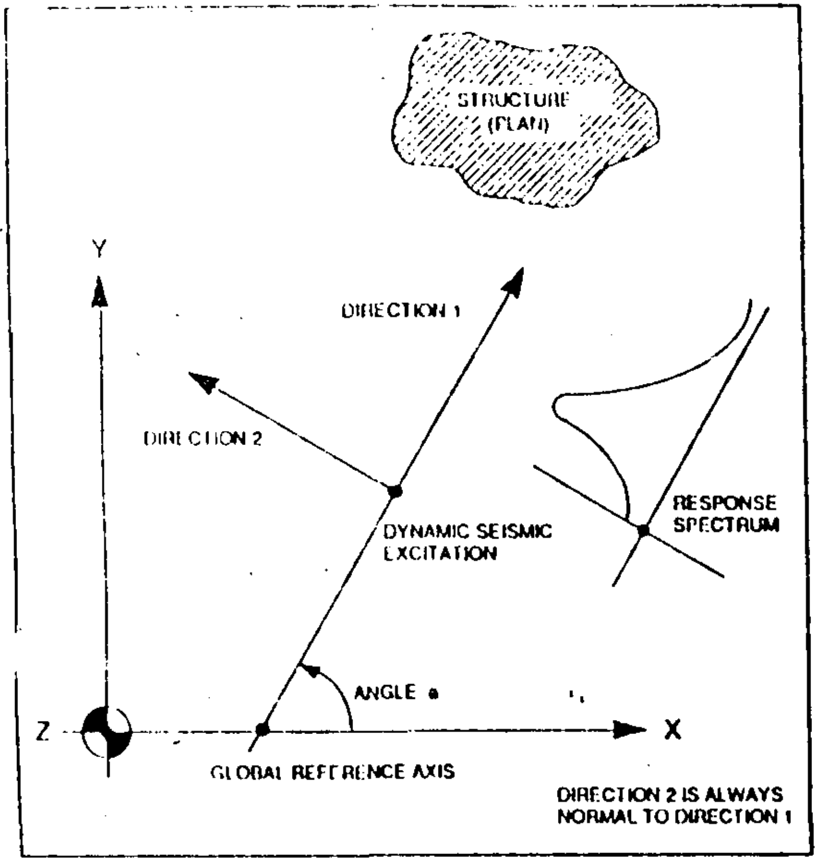


Figure X-28
Direction Convention for Dynamic Seismic Excitation

Absolute summation of the three directional components is possible instead by specifying

DIRC = 1

on the control information line (see Format Section b).

2. The response spectrum scale factor, s, is a multiplying factor that is applied to all of the spectrum values, s_1, s_2 and s_3 that are input in the spectrum curve data below. This

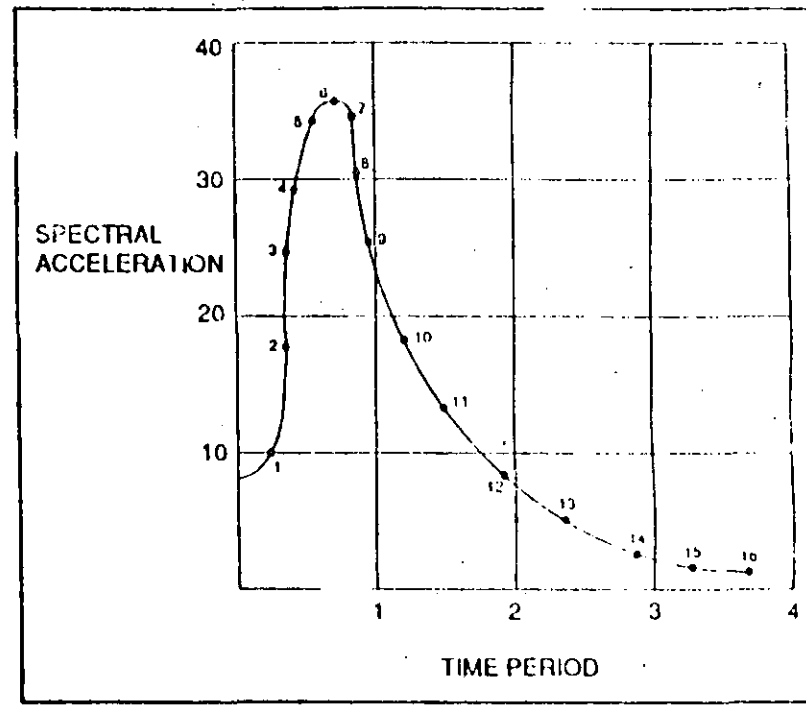


Figure X-29
Typical Response Spectrum Curve

factor is used for amplifying or reducing the spectrum intensity, or for transforming the spectral acceleration to consistent units (length/(time)^2).

3. The damping ratio, d, measured as a fraction of critical damping, must be a positive number less than 1.0. This factor is used in the complete quadratic combination (CQC) technique, per Reference [25]. If **d** is not specified, it is assumed to be zero, in which case the complete quadratic combination technique defaults to the square root of the sum of the squares (SRSS) technique.

4. The spectrum curves are defined by digitized points of time period versus spectral acceleration. The three entries s_1, s_2

and s_Z are the spectral acceleration values corresponding to Directions 1, 2 and Z, respectively, at time period t_p. See Figure X-29.

If excitation along a particular direction is not desired, the corresponding entry must be 0.

The time period values, t_p, (typically in seconds) and the spectral acceleration values s_1, s_2 and s_Z, must be in consistent units.

The values for t_p must increase numerically on successive data lines. All time periods and spectral values must be positive numbers. The range of t_p must cover the range of time periods associated with the lowest and highest modes being considered.

Only one set of concurrent ground acceleration excitation assignments is possible in any one run.

18. "TIMEH" Data Block

This data block is for defining the data associated with the dynamic time-history analysis.

Skip this data block if a dynamic time-history analysis of the structure is not desired. Otherwise prepare data for Format Sections a, b, c, d and e as described below. This data block requires that **nfq** or **nritz** be specified in the SYSTEM data block.

If both this and the SPEC data block are present in the input data, this block is ignored.

FORMAT

a. Separator

Provide one data line for the TIMEH separator in the following form:

TIMEH

b. Time History Control Information

Provide one data line in the following form:

ATYPE=**atype** NSTEP=**nstep** DT=**deltat**
NF=**nfunct** NV=**nv** D=**d**

c. Modal Damping Data

The damping values to be used for the different mode shapes (Ritz vectors) are specified in this data section. Skip this data section if a constant damping for all mode shapes is specified in Format Section b above, i.e., if **d** is not zero. Otherwise prepare this data section by specifying damping values, five pairs per line, for the first **nv** mode shapes (Ritz vectors) as follows:

$n_1\ d_1\ n_2\ d_2\ n_3\ d_3\ n_4\ d_4\ n_5\ d_5$
$n_6\ d_6 \ldots n_{10}\ d_{10}$
$n_{11}\ d_{11} \ldots$
$\ldots n_{nv}\ d_{nv}$

d. Function Definition Data Sets

Provide one set of data for each of the **nfunct** function definitions. Each function-definition data set consists of a control line immediately followed by the values of the function as shown below:

(i) Function Definition Control Line

Provide one data line in the following form:

NF=functn NPL=npl DT=dt
NAM=filename PRIN=pflag

(ii) Function Values

Use as many data lines as necessary to define the function values. The form of the function values data line depends on the function type:

If **dt** is specified, i.e., the function values are specified at equally-spaced time intervals, then provide **npl** values of the function per data line as follows:

$f_0\ f_1\ f_2\ f_3 \ldots f_{npl-1}$
$f_{npl} \ldots$
...

If **dt** is *not* specified, i.e., the function values may be specified at unequally-spaced time intervals, then provide **npl** *pairs* of values (of time and function) per data line in the following form:

$t_0\ f_0\ t_1\ f_1\ t_2\ f_2 \ldots t_{npl-1}\ f_{npl-1}$
$t_{npl}\ f_{npl} \ldots$
...

e. Load Function Assignment Data

Provide as many data lines as required to assign time functions to the static load conditions or to base accelerations for time history analysis. **End this data section with a blank line.** Prepare each data line in the following form:

LC=lcn NF=nf S=s AT=arrival ANGLE=a

DESCRIPTION

Variable	Note	Default	Entry
Time History Control Information			
atype	(1)	[0]	Analysis type = 0 Transient = 1 Periodic
nstep	(2)	[0]	Number of output values
deltat	(2)		Time increment for output
nfunct		[1]	Number of different time functions defined in Format Section d
nv	(3)	[nfq] [nritz]	Number of mode shapes (Ritz vectors) to be considered in analysis
d	(4)	[0]	Damping ratio for all mode shapes (Ritz vectors). Must be less than 1.0
Modal Damping Data			
$n_1,n_2...$	(4)		Mode shape (Ritz vector) numbers (in ascending order)
$d_1,d_2...$	(4)		Damping ratios associated with different mode shapes (Ritz vectors). Must be less than 1.0

Variable	Note	Default	Entry
Function Definition Data Sets			
functn			Function identification number (in ascending order)
npl	(5)	[5]	Number of points defined per line in Format Section d(ii) for this function
dt	(5)		Time interval spacing for function values
filename	(6)		Optional name of file containing data required in Format Section d(ii)
pflag	(7)	[0]	Print flag for function values =0 Do not print values =1 Print values
$f_0,f_1,f_2...$	(5)		Function values at time $t_0, t_1, t_2...$
$t_0,t_1,t_2...$	(5)		Time values
Load Function Assignment Data			
lcn	(8)		Static load condition number being assigned time function (or base acceleration direction if negative)
nf	(8)		Time function number being assigned to static load condition lcn
s	(8)	[1]	Scale factor
arrival	(9)	[0]	Arrival time for this load
a	(8)	[0]	Angle between the 1-direction and the global X-axis

NOTES

1. Two types of time history analysis are possible, transient and periodic. For periodic analysis, the period of the cyclic function is assumed to be **nstep • deltat**.

2. The time span over which the time history analysis is carried out is given by **nstep • deltat**. Responses are calculated after every **deltat** time increment, resulting in **nstep** values for each output response quantity.

   Since exact modal integration is used in computing the response, numerical instability problems are never encountered and the time increment may be any sampling value that is deemed fine enough to capture the maximum response values. One-tenth of the time period of the highest mode is usually recommended; however, a larger value may give an equally accurate sampling if the contribution of the higher modes is small.

3. It is possible to use fewer mode shapes (Ritz vectors) than calculated by the program. If **nv** is specified, it must be less than or equal to **nfq** or **nritz**.

4. It is possible to define either a single damping ratio for all mode shapes (Ritz vectors), or different damping ratios for each mode. If the damping ratio for all the modes is the same, the damping ratio can be given with the time history control information. If the damping ratios for the modes are different, they should be specified as pairs in Format Section c.

5. Two methods are allowed in the program for specifying time function data.

If **dt** is specified, the program will read function values (only) at equal intervals along the time axis at the rate of **npl** values per line. The corresponding time values for this method of input are taken as **0•dt, 1•dt, 2•dt** ....

If **dt** is not specified, the program expects to read *pairs* of time and associated function values at the rate of **npl** pairs per line. The time points **t0, t1** ... must be in numerically increasing order, but do not have to be equally spaced. Also if the first time **t0** is not zero, all input time points are modified by the program by subtracting **t0** from them.

If any function defined does not span the full time span **nstep • deltat** of the time-history analysis, the value of the function is taken as zero from the end of the function definition to time **nstep • deltat**.

6. If **filename** is not specified the program expects to read the function data of Format Section d(ii) in the input data file immediately after the function control line, Format Section d(i).

7. The function values and corresponding times are not echoed in the output file if **pflag** is left as 0.

8. The time varying loads are defined as:

$$R(x,t) = s \cdot F(x) \cdot T(t)$$

The user defines **F(x)** as one of the static load conditions, number **lcn**, and **T(t)** as one of the time functions, number **nf**, multiplied by a scale factor **s**.

Using negative values of **len** will allow the time functions to be applied as base accelerations as follows:

len= -1	Function is applied as a base acceleration in the 1-direction at angle a with the global X-axis.
len= -2	Function is applied as a base acceleration in the 2-direction at angle a+90° with the global X-axis.
len= -3	Function is applied as base acceleration in the global Z-direction.

See Figure X-28 for direction conventions.

Any number of different time varying loads can be defined in a single run either with or without simultaneous base motions. However, only the combined responses are output.

9. Different arrival times may be specified for each time varying load. The value of the loads from time zero until the arrival time is taken as zero. Arrival times may only be specified for transient analysis, i.e., **atype** = 0.

19. "COMBO" Data Block

This data block is for defining load combinations for the joint displacements and reactions, and element forces (or stresses).

Load combinations are defined as linear combinations of the previously defined basic **nld** load conditions and the dynamic (response-spectrum) load condition.

Skip this data block if no load combinations are to be generated. Otherwise, prepare data for Format Sections a and b as described below.

If this data block is not defined, output associated with each of the basic **nld** load conditions and the dynamic load condition will be created, with no combinations.

FORMAT

a. Separator

Provide one data line for the COMBO separator in the following form:

COMBO

b. Combination Data

In this data section, provide one data line for each loading combination required. **End this data section with a blank line.** Prepare the data lines in the following form:

i C=c_1, c_2, ... c_{nld} D=d

EXAMPLE

```
COMBO
1  C=1.4,1.7,0
2  C=1.4,1.4,1.4   D=1.4
```

DESCRIPTION

Variable	Note	Default	Entry
i	(1)		Combination identification number
c1, c2, ...	(2)		Multiplier for the static load conditions
d	(2)		Multiplier for the response-spectrum dynamic load condition

NOTES

1. There is no limit on the number of load combinations. The combination identification numbers must be in ascending, consecutive numerical sequence starting with one (1).

2. A load combination is defined as the sum of the **nld** static load conditions, multiplied by **c1, c2, ... cnld**, respectively, plus the dynamic load condition multiplied by **d**.

   The contribution from the response spectrum dynamic load condition will be zero if the SPEC data block is not defined.

20. "ENVELOPE" Data Block

This data block is for defining **envelope combinations** of the basic **nld** static load conditions and the dynamic (response-spectrum) load condition with the moving load cases from the SAP90 Bridge Analysis Module [12]. This data block may also be used without bridge moving load cases.

Each envelope combination produces maximum and minimum FRAME element force results due to the combined loads. If more than one envelope combination is defined, a total envelope of all envelope combinations is automatically computed. These results are printed and may be graphically displayed using program SAPLOT [5].

If this data block is present, results produced for the FRAME element will be for the envelope combinations. If this data block is omitted, the FRAME element results are for load combinations when the COMBO data block is present, or else for the load conditions. If only static load conditions are combined in the ENVELOPE data block, the results are the same as using the COMBO data block, except that a total envelope of all the combinations is produced.

The ENVELOPE data block does not apply to results for the joints or for the SHELL, ASOLID or SOLID element types. These results will always be for the load combinations when the COMBO block is present, or else for the load conditions.

Skip this data block if no envelope combinations are to be generated. Otherwise, prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the ENVELOPE separator in the following form:

ENVELOPE

b. Envelope Combination Data

Provide one data line in this section for each envelope combination required. **End this data section with a blank line.** Prepare each data line in the following form:

i C=c_1, c_2, ... c_{nld} D=d B=b_1, b_2, ..., b_{nc}

EXAMPLE

In this example, suppose that the first two static load conditions are dead load and transverse wind load, respectively, and that a dynamic response-spectrum analysis has been performed. The first envelope combines the dead load with dynamic load, automatically accounting for both positive and negative senses of the dynamic load. The last two cases combine the dead load with the wind load acting in two opposite directions. A total envelope will automatically be produced which is the envelope of these three envelopes.

```
ENVELOPE
1  C=1      D=1
2  C=1,1
3  C=1, 1
```


DESCRIPTION

Variable	Note	Default	Entry
i	(1)		Envelope combination identification number
c_1, c_2, ...	(2)	[0]	Multipliers for the static load conditions
d	(2)	[0]	Multiplier for the response-spectrum dynamic load condition
b_1, b_2, ...	(2)	[0]	Multipliers for the bridge moving-load cases

NOTES

1. There is no limit on the number of envelope combinations. The envelope combination identification numbers must be in ascending, consecutive numerical sequence starting with one (1).

2. Each envelope combination produces two results, a maximum and a minimum, for each FRAME element force response quantity.

   The maximum element force response for an envelope combination is the sum of the responses to the **nld** static load conditions multiplied by c_1, c_2, ... c_{nld}, respectively, plus the response to the dynamic load condition multiplied by d, plus the maximum responses to the **nc** moving load cases multiplied by b_1, b_2, ... b_{nc}, respectively.

   The minimum element force response for an envelope combination is the sum of the responses to the **nld** static load conditions multiplied by $c_1, c_2, \ldots c_{nld}$, respectively, minus the response to the dynamic load condition multiplied by **d**, plus the minimum responses to the moving load cases multiplied by $b_1, b_2, \ldots b_{nc}$, respectively.

The contribution from the dynamic load condition will be zero if the SPEC data block is not defined.

Normally only one of $b_1, b_2, \ldots b_{nc}$ should be non-zero in any single envelope combination to avoid multiple loading of the lanes. Moving load cases can only be included when the BRIDGE, VEHICLE and MOVING LOAD data blocks are present (See Reference [12]).

21. "SELECT" Data Block

This data block allows the user to selectively control the joint and element output produced by the program.

Skip this data block if no selective output control is required.

If this data block is *not* defined, displacements and reactions associated with *all* the joints will be output, and member forces and stresses associated with *all* the elements will be output.

If this data block *is* defined, then *only* the results associated with the joints and elements that are explicitly requested in this data block will be output.

If selective output control is required, prepare data for Format Sections a and b as described below.

FORMAT

a. Separator

Provide one data line for the SELECT separator in the following form:

SELECT

b. Joint/Element Selection Data

In this data section, provide as many data lines as needed to define those joints and elements that are to be included in the output created by the program. Selection types, joint and element identification numbers may be repeated and re-specified. The last specification given will be used. **End this data section with a blank line. Prepare the data lines in the following form:**

NT=nt ID=i_1, i_2, inc SW=isw

EXAMPLE

To produce displacement output for Joints 11 through 37:

NT=1 ID=11,37 SW=1

To produce member force output for Frame elements 25 through 45:

NT=5 ID=25,45

DESCRIPTION

Variable	Note	Default	Entry
nt	(1)	[pv]	Selection type
i_1	(2)		First joint or element number
i_2	(2)	[i_1]	Last joint or element number
inc	(2)	[1]	Joint or element number increment
isw	(3)	[pv]	On/off flag = 0 Exclude joint or element = 1 Include joint or element

NOTES

1. The selection type, **nt,** determines the type of output quantity being specified by this data line. The value of **nt** can be between 1 and 8 as follows:

	Joint number selection for:
nt= 1	Displacements
nt= 2	Reactions and Applied Loads
nt= 3	Mode Shapes
nt= 4	Averaged Finite Element Joint Stresses (SOLID Elements only)
	Element number selection for:
nt= 5	FRAME Elements
nt= 6	SHELL Elements
nt= 7	ASOLID Elements
nt= 8	SOLID Elements

16. **Bathe, K. J.**

Finite Element Procedures in Engineering Analysis, Prentice-Hall, Englewood Cliffs, N.J. (1982).

17. **Bathe, K. J. and Wilson, E. L.**

Numerical Methods in Finite Element Analysis, Prentice-Hall, Englewood Cliffs, N.J. (1976).

18. **Batoz, J. L. and Tahar, M. B.**

"Evaluation of a New Quadrilateral Thin Plate Bending Element," *International Journal for Numerical Methods in Engineering*, **18**, 1655–1677 (1982).

19. **Cook, R. D., Malkus, D. S., and Plesha, M. E.**

Concepts and Applications of Finite Element Analysis, 3rd ed., John Wiley & Sons, New York (1989).

20. **Hollings, J. P. and Wilson, E. L.**

"3–9 Node Isoparametric Planar or Axisymmetric Finite Element," *Report No. UC SESM 78-3*, University of California, Berkeley, California (Dec. 1977).

21. **Taylor, R. L. and Simo, J. C.**

"Bending and Membrane Elements for Analysis of Thick and Thin Shells," *Proceedings of the NUMEETA 1985 Conference*, Swansea (Jan. 7–11, 1985).

22. **White, D. E. and Hajjar, J. F.**

"Application of Second-Order Elastic Analysis in LRFD: Research to Practice," *Engineering Journal*, ACI, 28 (4), 133–148 (1991).

23. **Wilson, E. L.**

"A New Method of Dynamic Analysis for Linear and Non-Linear Systems," *Finite Elements in Analysis and Design*, **1**, 21–23, North-Holland (1985).

24. **Wilson, E. L. and Button, M. R.**

"Three Dimensional Dynamic Analysis for Multicomponent Earthquake Spectra," *Earthquake Engineering and Structural Dynamics*, **10** (1982).

25. **Wilson, E. L., Kiureghian, A. D. and Bayo, E. P.**

"A Replacement for the SRSS Method in Seismic Analysis," *Earthquake Engineering and Structural Dynamics*, **9** (1981).

26. **Wilson, E. L. and Tetsuji, I. J.**

"An Eigensolution Strategy for Large Systems," *Computers and Structures*, **16** (1983).

27. **Wilson, E. L., Yuan, M. W. and Dickens, J. M.**

"Dynamic Analysis by Direct Superposition of Ritz Vectors," *Earthquake Engineering and Structural Dynamics*, **10**, 813–823 (1982).

28. **Zienkiewicz, O. C., and Taylor, R. L.**

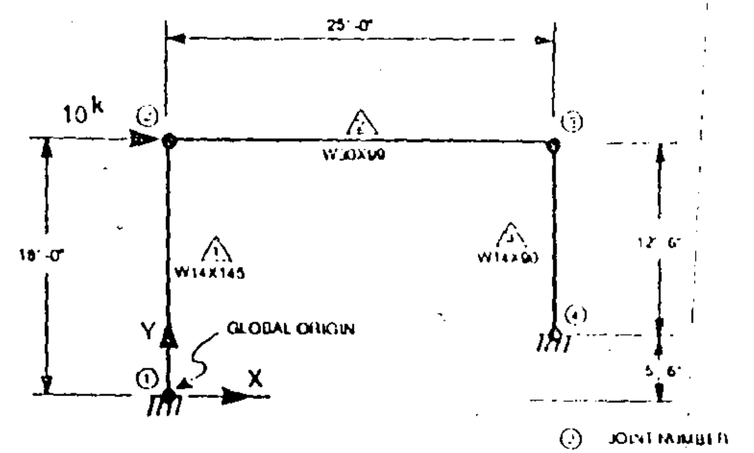
The Finite Element Method, 4th ed., McGraw-Hill, London, vol. 1 (1989); vol. 2 (1991).

DATA BLOCK	DESCRIPTION	MANDATORY
1. Title Line	Job Title Information	Yes
2. SYSTEM	Job Control Information	Yes
3. JOINTS	Joint Coordinates	Yes
4. RESTRAINTS	Joint Restraints	Yes[1]
5. SPRINGS	Joint Spring Supports	No
6. MASSES	Joint Masses	No
7. POTENTIAL	Joint Temperatures and Pressures	No
8. CONSTRAINTS	Joint D.O.F. Relationships	No
9. FRAME	Frame Element Data	Yes[2]
10. SHELL	Shell Element Data	Yes[2]
11. ASOLID	Asolid Element Data	Yes[2]
12. SOLID	Solid Element Data	Yes[2]
13. LOADS	Applied Joint Loads	No
14. DISPLACEMENTS	Applied Joint Displacements	No
15. PRESTRESS	Prestress Loading on Beams	No
16. PDELTA	P-Delta Analysis Control	No
17. SPEC	Response Spectrum Data	No
18. TIMEH	Time History Data	No
19. COMBO	Load Combination Data	No
20. ENVELOPE	Envelope Combination Data	No
21. SELECT	Selective Output Requests	No

[1] This data block may be omitted only if the structure is fully supported by springs.

[2] At least one of these four data blocks must be present in the data file.

Figure VII-1
SAP90 Data Blocks



(a) SAP90 MODEL

```
SAMPLE PORTAL FRAME EXAMPLE
SYSTEM
L = 1                                          : Number of load conditions
RESTRAINTS
1 4 1        R = 0,0,1,1,1,0                   : All parts are in the X-Y plane
1 4 3        R = 1,1,1,1,1,1                     Fix base joint
                                               : Blank terminator
JOINTS
1            X =   0.0     Y =   0.0           : Coordinates for joint 1
2            X =   0.0     Y =  18*12          : Coordinates for joint 2
3            X =  25*12    Y =  18*12          : Coordinates for joint 3
4            X =  25*12    Y =  6.5*12         : Coordinates for joint 4
                                               : Blank terminator
LOADS
2  L = 1        F = 10,0,0                     : Load at joint 2
                                               : Blank terminator
FRAME
NM = 3                                         : Number of section properties
1   SH = W14X145            E = 29500          : Section property data
2   SH = W30X99                                : Section property data
3   SH = W14X90                                : Section property data
1   1   2       M = 1       LP = 1,0           : Data for element 1
2   2   3       M = 2                          : Data for element 2
3   3   4       M = 3                          : Data for element 3
                                               : Blank terminator
```

(b) SAP90 INPUT DATA

Figure VII-2
Typical SAP90 Input Data

Details of the SAP90 input data options are presented in Chapter X.

The contents of a typical input data file is shown in Figure VII-2.

B. Free Format

All SAP90 input data is prepared in free format. In other words, data on a particular data line does not have to correspond with prespecified column locations. The data is input as lists of data items *separated by a comma or by one or more blanks*. The data items may be numbers or alpha-numeric strings. All alphabetic characters that appear in the input data may be uppercase or lowercase.

The free-format data is of two forms:

(1) List-directed input

(2) Nonlist-directed input

List directed input is a free-format list of data items preceded by an identifier (and an equal sign), such as

XI=2000,4501

No blanks can separate the identifier from the equal sign, or the equal sign from an alpha-numeric string.

Nonlist-directed input is just a list of data items with no preceding identifier, such as

A typical data line may be a combination of list-directed and nonlist-directed input, such as

20,45,5 XI=1200,1500 AV=15,25

The nonlist-directed input must always appear as the first list of data on the data line, but the list-directed input can appear in any sequence. In the above example 20,45,5 must be first, but the XI=1200,1500 sequence can be before or after the AV=15,25 sequence. If a sequence is only partially entered, the trailing (omitted) items are taken as zero. For example, XI=1200 is the same as XI=1200,0.

Decimal points for whole floating point numbers are not necessary. For example, the number (6.0) may just be entered as (6). Scientific exponential notation is also allowed. For example, the number 1.5×10^{10} may be entered as 1.5E10.

The data file must not contain any spacing tab characters. Some text editors automatically insert tab characters in text files by default.

C. Comment Data

Any line (except the first line) having the letter C in column 1, and a blank column 2, is treated as a comment line and is ignored by the program. A colon (:) indicates the end of information on any line. Information entered to the right of the colon is also ignored by the program.

OUTPUT FILENAME	FILE CONTENTS
1. EXAMPLE.SAP	Tabulated Input Data Echo
2. EXAMPLE.EQN	Optimized Equation Numbers
3. EXAMPLE.EIG	Frequencies and other Modal Parameters (Using Eigenvectors)
4. EXAMPLE.RIT	Frequencies and other Modal Parameters (Using Ritz vectors)
5. EXAMPLE.SPC	Spectral Analysis Modal Factors
6. EXAMPLE.SOL	Joint Displacements and Reactions (Static and Dynamic)
7. EXAMPLE.F3F	FRAME Element Forces (Static and Dynamic)
8. EXAMPLE.F4F	SHELL Element Forces (Static and Dynamic)
9. EXAMPLE.F5F	ASOLID Element Stresses (Static and Dynamic)
10. EXAMPLE.F8F	SOLID Element Stresses (Static and Dynamic)
11. EXAMPLE.FEF	Element Joint Forces (Static and Dynamic)
12. EXAMPLE.ERR	Solution Errors and Warnings

Figure VIII-1
Output Files Associated with Input Data File "Example"

To review any of these files on the terminal, the user may use the MS-DOS **TYPE** command or a text editor.

To print an output file, the MS-DOS **PRINT** command may be used. Appropriate page ejects and line counts are built into the files.

B. Contents of Output Files

Sample output files are presented below corresponding to a sample input data file named EXAMPLE as shown in Figure VIII-2.

B.1. File EXAMPLE.SAP

This output file contains a tabulated echo of the information from the data blocks existing in the input file.

This file is created by the SAP90 preprocessor and contains details of any errors that are detected during the preprocessing phase.

Before executing the **GO** command, this file must be printed and checked for numerical and compatibility errors.

The file also contains a list of the output files that will be created by executing the program. This output file is always created. See Figure VIII-3.