INTRODUCTION

The four elements to the CADRE Pro documentation and tutorial are the

- CADRE PRO USER MANUAL
- HELP AND THE “GETTING STARTED” TUTORIAL IN HELP
- THE STATIC AND DYNAMIC DOCUMENTED SAMPLE MODELS
- WHITE PAPERS ON SPECIAL SUBJECTS – AVAILABLE ONLINE

The sample models are supplied and installed with CADRE Pro. Each sample has a Notes page available from Edit/Information or by clicking on the file name on the menu bar. This is the same notes page you can use to document the models for your own projects.

These sample models provide the main benchmark validation and confirmation for CADRE Pro since many of the samples are simple enough to be solved by classical techniques and in many cases classical solutions are also provided.

This guide to the sample models presents each model supplied with CADRE Pro along with the descriptive information on the model’s notes page. In some cases the information here is identical to the notes page but in many cases, additional information such as enhanced diagrams and pictures are provided in this presentation.

Some of the models provided are the same as the ones developed by the Help, “Getting started” tutorial. In those cases, only an abbreviated discussion is provided in this guide and the reader is referred to the exercise in Help. In other cases, the models are described in much greater detail to illustrate specific methods and techniques.

The sample models are presented here in simple alphabetical order using the model file name, NOT in order of difficulty or complexity.

STATIC SAMPLE MODELS

AFRAME.FEM (offset loads with eccentric beams)

Usually A-frames can be simply constructed with standard beams modeled along the centerlines of the primary members. However, there can be configurations that may induce significant eccentric loads in the columns. This is an example with a cross beam pinned to extended brackets on the sides of the columns.

An off-center pin connection can apply the vertical load directly into the hangers creating a significant end moment. While one could use a short stub beam for the attachment bracket, another method of handling this issue is to use the 'Eccentric beam' element to model the columns. This alternative method can sometimes lead to improved numerical conditioning over the stub beam approach.
This sample model demonstrates:

- the use of ‘Y Offset’ style Eccentric beams
- discretizing eccentric and Pinched beams
- setting pin factors directly on the model
- stability with eccentric beams

DESCRIPTION

Dimensions are shown on the diagram above.

Columns are steel square tube 4 x 4 x 0.125.

Cross beam is steel rectangular tube 7 x 4 x 0.125 and pinned at each end to the column brackets.

The columns are eccentrically loaded 7 inches off center at the upper end.

The load of 24180 pounds is distributed across 78 inches of crossbeam (310 lb/in).

The columns are fully fixed at ground.

CONSTRUCTION

Instead of setting nodes and drawing individual beams you can use ‘Utilities/Quick modeler’.

Set: ‘Wire mesh’ in the XY plane

Enter corner points: (0, 0, 0) and (92, 165)

Enter divisions: 1 and 1 (this makes a simple rectangle)

Open the Nodal Editor mode

- relocate the upper left node (#3) 7 inches to the right (i.e. Select it, right-click, and use Set X = 7)
- relocate the top right node (#4) 7 inches to the left (i.e. Select it, right-click, and use Set X = 85)
Open the Element Editor mode.

-delete the bottom line between #1 and #2 (i.e. Select it, right-click, and use **Delete elements**)

![Diagram of elements](image)

**ORIENTATION**

All members are oriented in-plane (XY plane in this case).
The cross beam is oriented to lower left node.
The left column is oriented to lower right node.
The right column is oriented to lower left node.
The columns should be directed (origin to axis node) in the same direction.
You can select them and use **Show direction** to verify.

![Diagram showing orientation](image)

If re-directing of a column is needed, select it, and use **Tools/Swap elements nodes**.
LIBRARY PROPERTIES (INITIALLY)

(1) Crossbeam: Rectangular tube 7 x 4 x 0.125 imported from the US Steel database

This crossbeam will be configured later with pinned ends but for now, leave it as a 'Standard beam'.

(2) Columns: Square AISC tube 4 x 4 x 0.125 imported from the US Steel database

Define this entry as 'Eccentric beam'

Set: Style = Y Offset (this style allows Y offsets with each end having different values)

Note: Y is the beam direction toward or away from its reference node.
Set Origin end: Dyo = 0
Set Axis end: Dya = 7

This assumes the columns are directed (origin to axis node) from bottom to top!
Otherwise you would reverse the Dyo and Dya values.

ASSIGN PROPERTIES

Open the Element Editor mode.
Select both vertical column members and assign the 4 x 4 x 0.125 square tube properties to them.
Select the cross beam and assign the 7 x 4 x 0.125 rectangular tube properties to it.
The columns should take on the appearance of the eccentric beam schematic at this point.

GROUP NAMES

Open the Element Editor mode.
Select both column members, right-click and use 'Apply group name' and name them Column.
Select the cross beam and name it Crossbeam.

DISCRETIZE

Open the Element Editor mode.
Select both of the columns, right-click, and use Divide elements. Divide into 10 segments.
Select the cross beam member, right-click, and divide into 6 members.
The divide method will gradually taper the offset (distance from load path to neutral axis) of the column segments from 7 inches at the top to near 0 at the bottom.

Notice that the nodes follow the assumed load path and taper inward to eventually meet the cross beam, but the elastic or neutral axis of the columns remains vertical.

The element library has been automatically augmented with entries to support these new offset segments.

PINNED ENDED CROSSBEAM

The cross beam is still fully fixed at the ends.

You could have initially set it up as a Pinned beam with Z pins (O-Zpin = 0 and A-Zpin = 0) in the library and it would be finished at this point because the smart divide feature would have correctly configured the end segments leaving the intermediate segments as fully fixed to each other. However, here we will demonstrate the alternate method of directly setting pins on the model itself which can be more intuitive.

Select right end segment of the crossbeam (ensure that only one single element is selected and nothing else!). You can use Show direction to verify that the left end of this segment is the origin node and the right end is the axis node.

With the segment selected right-click and use Set beam pin factors from the pop-up menu or use the same item under Tools. Pin the end with a rotation axis normal to the plane of the frame (local Z axis of the element in this case).

Check Z pin on the Axis end. Set it to zero and note the axis is shown on the dialog icon as well as on directly on the model. You can apply pins easily by trial and error since the proposed result is shown on the model. If not what you expect, change the setting and try again.
Although shown on the model, the changes do not actually take effect until you use **OK** to leave the dialog.

Select **OK** and the pin is set along with a temporary indicator confirming the pin axis that you just set.

Perform the same step on the left end of the left segment of the cross beam.

To review the pin axes at any time, just select the entire crossbeam, right-click and use **Show beam pin axes.**
AT THIS POINT THE ELEMENT LIBRARY HAS BEEN GREATLY EXPANDED TO INCLUDE AN ENTRY FOR EACH DIFFERENT ECCENTRIC BEAM AND EACH TYPE OF PINNED BEAM AS WELL AS THE ORIGINAL STANDARD BEAM ENTRY.

RESTRAINTS

Open the Nodal Editor mode
Select the two ground points and set fully fix in all 6 degrees of freedom.
In addition, if you only want to consider lateral stability only (2D analysis), restrain some points in Z (e.g. fix the 2 crossbeam end nodes in the Z direction only).

LOADS

Select all the cross beam segments (78 inches). You can use Ctrl_G then choose Crossbeam group to select them.
Right-click and use Set loads.
Open the Projected tab and set:
Y Negative
Replace existing
Intensity = 310 lb/in, Replace
Use the Apply projected load button
Total load should be -24180 lb (downward)

Use **OK** to apply this load condition to the model.

**SOLVE**

You should be in the habit of always running the **Bandwidth manager** after completing a new model. It won’t matter much with small models like this but may still provide a faster solution.

Use **Solve/Static/Advanced**.

Set just 1 increment

Use **Results/Beam stress maximums**

Stress Sn1 or Sn2; Maximum of either end

<table>
<thead>
<tr>
<th>Group name</th>
<th>Column</th>
<th>Crossbeam</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>29703</td>
<td>46261</td>
</tr>
<tr>
<td></td>
<td>-42464</td>
<td>-46865</td>
</tr>
</tbody>
</table>

Select nodes 11 and 20 and use the measure button to check the distance between these nodes before and after solution.

Before solution the distance is 82.200. After solution the distance is 83.614. The difference (1.414 inches) is the expansion created by the end moments of the eccentric load.

With the 24180 lb this A-frame is loaded almost to collapse.

**THEORETICAL PURE COLUMN BUCKLING VALUE** (for a single column pole the pin factor is c = 1/4)

Ignoring offset,

\[ P_{CR} = \frac{c^2(2EI)}{L^2} = 12344 \text{ lb} \]
For a single side or 24688 total

Consider this to be an upper theoretical limit since the offset would reduce this figure somewhat if we used a more realistic beam-column formulation.

Reset the total load to 25000 pounds (Intensity = 320.513 lb/in). One way to revise quickly the load from 24180 to 25000 is to factor the current load by 1.033912 using Set loads with the 'File tab', and then use the Factor button.

With the total load at 25000, resolve again with advanced 1 increment.

Instability should be detected.

Since a node (rather than an element) is flagged (i.e. selected and highlighted) as unstable then the entire structure is probably unstable. The identified node is just the first node found along the diagonal of the model stiffness array that is less than or equal to zero. Many, if not most, other nodes are likely unstable as well.

You can re-solve using 1000 increments starting at increment 950 to home in on the exact buckling load in percent of applied load.

Unstable at just above 98.0% of 25000 = 24500lb

The reduction below the pure column buckling value of 24688 is the additional instability from the offset moment.

**AMPHITHEATER.FEM**

This is a structural steel model of an actual construction project of an amphitheater with fabric cover over bar joists.

![AMPHITHEATER.FEM](image)

This model was created from an imported CAD DXF file then carefully configured for analysis.

While it may seem more convenient to draw a model in CAD and import to CADRE Pro, the real work begins there. Every element must be properly connected, oriented, and configured with correct properties.

When constructed within CADRE Pro, one can ensure these items are established as the model is built up and these settings carry onward as like components are replicated and merged to build the final model.

**ANTICLAS.FEM (ANTICLASTIC BEHAVIOR)**

This model is a Lexan (polycarbonate) flat sheet 400 x 400 mm x 8 mm with pure torsion applied equally and oppositely on opposing edges. It is modeled in the Newton-mm (Mpa) system.
Lexan 9034 Polycarbonate
E = 2378 Mpa
Poisson (V) = 0.37
Thickness (T) = 8.0 mm

This sample is intended to demonstrate the biaxial effect of stress and strain in anticlastic behavior (i.e. plate has two curvatures of opposite signs).

Stresses and strains are relatively constant over the entire surface. Increase the 'exaggerate' control to make the saddle more visually pronounced.

Poisson's ratio is a measure of the effect of one stress axis on the other. Try reducing the Poisson ratio to zero in the element library, and then increasing it up to 0.99 to demonstrate that the saddle effect is primarily controlled by the Poisson ratio.

The opacity setting for the Lexan is set to 50% in the Element Library in order to improve the realistic view of the deformed transparent sheet.

The restraint provided in the center is to prevent rigid model instability. All FEA must be restrained against free body movement in order for a solution to take place even when loads tending in those directions are zero.

The restraint, in this model, is carefully placed and configured so that it is not actually restraining the natural behavior of the model. This is confirmed by examining the reaction forces at the restraint node after solution. They are zero or very nearly so (on the order of 1E-10!)

**ARCH60.FEM (Curved beam)**

**LOADED WITHIN THE PLANE OF CURVATURE**

This is an example taken from Roark's *Formulas for stress and strain* section on curved beams and arches (Fifth Edition, article 8.3). This sample model involves using the Basic Shapes generator for non-standard sections and also the use of the utility for inserting arcs and circles.

Also: See the example CurvedBeam.fem which illustrates a curved beam loaded normal to the plane of curvature.
It is a 50 inch radius arch spanning 120 degrees (i.e. from 30 to 150 degrees from the right horizontal axis).

The left reaction is a horizontal slider that is fixed in vertical displacement but allowed to slide horizontally and also to rotate freely as needed. The right restraint is fully fixed in all 6 degrees of freedom.

The load (both a force and a moment) is applied at a point 110 degrees from the right horizontal.

ARCH CONSTRUCTION

Start a new model

- Extents: -45, 0, -5; 45, 50, 5
- Grid: 5

The center and end points at:

\[
X = 0; Y = 0 \\
X = 50(\cos 150) = -43.30127; Y = 50(\sin 150) = 25 \\
X = 50(\cos 30) = 43.30127; Y = 50(\sin 30) = 25
\]

In the nodal editor, set three points:

- Node 1 (Center) 0, 0, 0
- Node 2 (Left leg) -43.30127, 25, 0
- Node 3 (Right leg) 43.30127, 25, 0

For convenience, you can turn on the snap tool and snap to -45, 25, 0 and 45, 25, 0 and then reset the X dimension exactly by selecting the node, right-clicking, and using 'Set X'.

The model is constructed with **Utilities/Insert structures/Circles and arcs**. Select the center then the left end and the right end (in order).

Enter 60 segments (Hint: choose a convenient value that will place a node at the 110 degree load point)

Choose Arcs < 180

Check the 'Attach nodes' option

ELEMENT PROPERTIES

The section is a "T" section made of steel with E= 3E+07 and G=1.2E7.

From the element library, the section properties are set up with CADRE Pro's "Basic Shapes" property generator tool on the Library dialog.

This brings up the basic shapes module which can generate the properties for most standard shapes such as the "T" section.

The following dimensions closely match the Roark problem statement.
T-section d=4 x b=4 x Tw=0.23 x Tf=0.254; Ri = 0.3125; flat flange.

Resulting in these properties

\[
\begin{align*}
\text{Area} & = 1.9195 \times 10^0 \\
I_y \text{ (Inertia)} & = 1.36004 \times 10^0 \\
I_z \text{ (Inertia)} & = 2.89931 \times 10^0 \\
J \text{ (Torsion coefficient)} & = 4.71292 \times 10^{-2}
\end{align*}
\]

Import (or fill in) the material and section properties and exit the element library with OK.

**IN THE ELEMENT EDITOR MODE**, **SELECT AND ASSIGN THE SECTION PROPERTIES TO THE ENTIRE ARCH.**

**LOAD POINT**

The load point, P is located on the arc at 110 degrees from the right horizontal (i.e. 90 + 20).

\[
\begin{align*}
X & = 50 \cos 110 = -17.101 \\
Y & = 50 \sin 110 = 46.98
\end{align*}
\]

With the strategic choice of 60 segments on the 120 degrees of arc (2 degrees per segment), this happens to be precisely at the 10th node (node 23) counterclockwise from the peak (node 33).

You can select the center node at (0, 0), then node 23, then node 33 (at the top), and use the measure tool (with the Ctrl key depressed) to verify the angle is precisely 20 degrees from the top.
If there were no node exactly at the load point, you could easily make one. You could just select the closest and reset its coordinates to the X, Y values for the load point. Or, perhaps divide the segment between the two closest nodes in half creating a new node there, then relocate that node to the X, Y value of the load point. Some strategic setting of the node spacing at the outset can save work later.

**NODE NAMES**

The model here is shown set up with user defined names for relevant nodes (RF, R1, R2 for the initial set points and P for the load application point). While still in the nodal editor you can choose these points, one at a time, and rename them. All other nodes were selected and hidden to provide clarity.

**REACTIONS**

Set R1 (free, fixed, fixed, fixed, fixed, free) as a horizontal pinned slider
Set R2 (fixed, fixed, fixed, fixed, fixed, fixed) as a fully fixed in all 6 DOF

**LOADS**

The applied loads (Fx = + 1000 and Mz = - 8000) are entered by selecting the node P and using the ‘Load’ button in the Nodal Editor or ‘Set Loads’ from any pop-up menu, or ‘Edit/Loads’ from the main menu bar.

- Fx = 1000 lb
- Mz = - 8000 in-lb (by right hand rule a clockwise moment in the XY plane as shown is negative)

Turn the load arrows on to see the applied force arrow (single head) and applied moment arrow (double head). Roll the model with Rot X to better view the double headed moment arrow.

Solve, then select left reaction node R1 and use the Eye view tool 🌍 to read some relevant results:

- Lateral displacement (X) of left end 0.04804
- Vertical reaction (Ry) of left end is -206.2 pounds

Roark gives the classical answers as 0.0472 and -206.2.

However, the classical Roark solution ignores the small deformation due to axial compression in the arch. If you desired to simulate this assumption, then increase the area of the beam by about 5 times leaving all other stiffness properties the same.

For example, set A = 10 in the element library and resolve:

- Lateral displacement (X) of left end 0.0473
- Vertical reaction (Ry) of left end is -206.2 pounds
Although one can re-create the limiting assumption of the classical analysis, the FEA solution here is actually the most accurate. This is not always the case and very often classical solutions are more precise than the FEA solutions.

**BASEPLATE.FEM (Post base plate on an elastic concrete foundation)**

**THIS BASE PLATE MODEL IS THE SUBJECT OF WHITE PAPER NO. 010.**

There are ample guidance in the AISC manual and other specifications in order to handle base plates by classical methods and those methods are generally preferable for structural substantiation since the material allowables and design factors are more or less tuned to those particular methods. However, for investigative purposes one may want to perform a detailed analysis of these components.

Some discussion on setting up loads for such models follows below. See the white paper #10 for much more detail of construction and alternate loading cases.

The white papers are available at [https://www.cadreanalytic.com](https://www.cadreanalytic.com) for evaluators as well as licensed users.

The baseplate is a steel plate 9 x 9 x 0.25 inch with eight 9/16 inch diameter holes for 1/2 inch bolts. It supports a 4 x 4 x 0.25 post (footprint) with a 3/16 inch weld.

This model demonstrates:

- Hole meshing utility
- Copy and paste elements utility
- Setting foundation stiffness
- Setting one way restraints
- Using the Hydrostatic loading method to represent an applied moment

**LOADS**

As provided in the sample files, the model is loaded with a pure overturning moment on the post footprint. That load is developed using a load line artifice along with the hydrostatic loading method which can apply a linearly varying load. This type of load is easily applied using the hydrostatic loading feature which treats linearly varying loads. The load is applied to a line of beam elements defined as "Load line" representing the footprint of the post.

1) Select the "Load line" elements
2) Right-click and use **Set loads** and open the Hydostatic tab.
3) Direction of increasing load is X; **Negative**
4) Apply in **Both** directions (i.e. in +X and in -X)
5) Direction of the load vectors is normal to a user defined system **System 1**. Choose **System 1**
6) **Replace** existing
7) **Apply to Beams only** (as selected)

The specified criterion is to apply a10000 in-lb pure moment about the Y axis of user **System 1**.

Try an intensity of 1.0 lb/in/in and use **Apply hydrostatic load**. Result >> My = 42.625

Next try 10000/42.625 = 234.604 lb/in/in. Result My = 10,000

**OK** to exit and the load condition is set as shown in the picture below.
Other loading cases (i.e. pull-up, push-down, and lateral) are demonstrated in white paper #10.

RESTRAINTS

That is 2257 boundary restraint conditions which include all nodes in the model. The foundation restraint is set up as one way spring restraints on 1745 nodes. In addition, there are 512 pinned restraints on nodes representing the bolt head and washer area.

This is one example of modeling a base plate that illustrates many construction and analysis techniques that can be applied to other types of components as well.

For details of construction, restraining, and loading with different loads, review the white paper No. 010.

SOLUTION

The solution iterates automatically until a stable condition of active and inactive one-way restraints is achieved. For the pure moment condition this takes about 5 repeats.

A color map of von Mises stress due to the pure moment is shown below.

Notice the free lift off from the foundation due to the one-way spring restraint but with resistance against push down due to the concrete spring resistance.

BEAM869.FEM (CONTINUOUS BEAM - RAMP LOAD)

This is an exercise taken from a Strength of Materials textbook. It is covered in more detail in Help “Getting started”, Exercise 25.

Turn on the force arrows and set them to the push type to see the load distributions.

(Note: You can go to Options/Settings, open the Model tab and set the Load Arrow sizes to Max = 50% and Min = 1% and the load distribution will display much more clearly for this model.)
The beam is a 32 foot long AISC W12x30 steel beam.

The problem is to find the displacement at point 5 (midway between points 1 and 2). The textbook uses the classical "Three moment equation" to solve the problem. The classical text book answer is 0.0001492 feet downward.

The model will be constructed entirely with the foot pound system of units.

This example illustrates:
1) Using the conversion utility to set up units that match a problem statement
2) Using the global loader 'projected feature' to apply constant load distribution
3) Using the global loader 'hydrostatic feature' to apply a triangular load distribution
4) Using the discrete loader to apply discrete loads

Create the beam using the Quick Modeler assuming units of pound and feet
(0, 0, 0)
(32, 0, 0)
64 segments

Set the orientation node at point (0, 5, 0) just above the beam.

REACTIONS
Locate the 4 reactions points at 0, 10, 20, and 32 feet and set the bounds to 'pinned'. You can also select and revise one of the bound points to also include Xrot as fixed just to ensure good stability against free rotation about the X axis (Not essential for this problem, but a good habit to adopt).

Doing this now helps mark and identifies these locations for use in later operations.
SECTION PROPERTIES

Go to the Element Library. Use the section database and import an AISC W12x30 section from the USA Steel set of section properties. These are in the units of INCHES and pounds so must be converted in the next step.

Use **OK** to import the section. Then use **OK** to exit the Element library.

CONVERSIONS

You could have made the original model in inches and worked the whole exercise in lb and inches without further need for conversion, but here we want to illustrate the use of the conversion utility to convert parts of the model by custom conversions. The section properties in the library imported from the database are in inches while our model is dimensioned in feet. All dimensions must have the same root unit. Here we make the Element Library match the model by converting those items from inches to feet.

Go to **Utilities/Conversions**; check the box for **Custom conversions**.

Enter the following factors to convert ONLY these specific section properties for the model:

- Area, A: 0.00694444 i.e. \(1/(12^2)\)
- Inertias, \(I_y\) and \(I_z\): 0.00004823 i.e. \(1/(12^4)\)
- Stress section moduli, \(S_y\) and \(S_z\): 0.00057370 i.e. \(1/(12^3)\)
- Elastic moduli, \(E\) and \(G\): 144 i.e. \(12^2\)
- Unit section weight, W: 12

Other entries should be 1.0.
The last item will modify the unit weight ($W/L$) parameters in the library. The $W/L$ parameter is actually not used in this example since the beam is considered to be weightless.

ASSIGN PROPERTIES

SELECT ALL THE ELEMENTS. RIGHT-CLICK AND USE SET PROPERTIES. APPLY THE W12X30 SECTION TO THE ENTIRE BEAM.

LOADING

The best mode to use with this model for applying a distributed loading is the Element Editor mode so you can have nodes circles visible and still easily select elements with the frame selector.

1) Load the portion between 1 and 2 with the 100 lb/ft distributed load.

Drag a frame over all elements between points 1 and 21 to select all 20 of those elements

Right-click on the screen, choose Set loads. Open the Projected tab.

Select Y, Negative and Replace existing.

Enter 100 in the intensity field. Don't check Use compensation.

Use the Apply projected load button to apply the load.

Use OK to exit and keep this loading. The total load should be -100 x 10 or -1000 lb

2) Next apply 1/2 of the triangular part from point 21 to half way to point 41 (i.e. point 31).

Drag a frame over the 10 elements from point 1 to the half way node between 21 and 41 (i.e point 31).

Right-click on the screen, choose Set loads. Open the Hydrostatic tab.

Choose X and Positive for the direction of increasing load, SET MERGE WITH EXISTING

Choose 10 (i.e. X = 10) as the point of zero load. Choose to keep Pos (+) loads or Both.

In the intensity field enter 32 lb/ft per ft (i.e. 160/5 = 32 lb/ft per foot). Don't check Use compensation.

Use the Apply hydrostatic load button to apply the load. Accumulative total should be -1000 – 32x5²/2 = -1400 lb

Use OK to exit.

Note: The hydrostatic loader applies the load normal to the beam. Positive normal is the direction away from the side of the reference node which we set up above the beam. Therefore a positive intensity +32 lb/ft would be applied
normal to the beam and downward as desired. (If the beam orientation node was below, then the reverse would be true)

3) Next apply 1/2 of the triangular part from half way to point 31 to point 41.
Drag a frame over the 10 elements from the middle point 31 to point 41.
Right-click on the screen, choose Set loads. Open the Hydrostatic tab.
Choose X and NEGATIVE for the direction of increasing load. Set Merge with existing
Choose 20 (i.e. X = 20) as the point of zero load. Choose to keep Pos (+) loads or Both.
In the intensity field enter 32 lb/ft per ft (i.e. 160/5 = 32 lb/ft per foot). Don't check Use compensation.
Use the Apply hydrostatic load button to apply the load.
The accumulated load should be -1000 -400 -400 = -1800 lb.
Use OK to exit.

4) Apply the discrete loads on the last portion.
Close the Element Editor.
Select the two nodes at X = 23 and X = 29 feet (should be nodes 47 and 59).
Right-click and choose Set loads
Since nodes were selected you are taken to the discrete load assignment dialog.
Enter Fy = -400 pounds
Click on the Assign button.
Use OK to exit.
The beam should be fully loaded.

LOAD VECTORS

Turn on the load arrows from the toolbar to see the discretized load vectors applied to each node in the distributed spans. With default arrow settings, you can barely discern the triangular shape since the individual vectors are small (50 to 70 lb) compared to the large 400 pound discrete ones and so are less than the minimum arrow length (which is about 25% of the maximum arrow length by default). Above we have set the maximum to 50 and the minimum to 1 which helps the visualization but also makes for long arrows on some nodes.
Check under Model/Gross properties to confirm that the total load is 2600 pounds on the beam.
ASSIGN PROPERTIES
Select all the elements. Right-click and use Set properties. Apply the W12x30 section to the entire beam.

SOLVE
The FEA solution for displacement at node 11 (midway from point 1 to 2) is -1.43860E-04
The bending moment diagram can be displayed by selecting all elements and using Results/Load diagram and choosing Loads diagrams.
Select Bending Moment, Z.

Solution by the classical 3 moment equation provides the following result for comparison:

<table>
<thead>
<tr>
<th></th>
<th>3-moment</th>
<th>This model</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>M2</td>
<td>-979.51807</td>
<td>-975.81300</td>
</tr>
<tr>
<td>M3</td>
<td>-1081.92771</td>
<td>-1080.50000</td>
</tr>
<tr>
<td>M4</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>M5</td>
<td>760.24095</td>
<td>762.09300</td>
</tr>
<tr>
<td>R1</td>
<td>402.04819</td>
<td>402.41900</td>
</tr>
<tr>
<td>R2</td>
<td>987.71085</td>
<td>987.11300</td>
</tr>
<tr>
<td>R3</td>
<td>900.40160</td>
<td>900.51000</td>
</tr>
<tr>
<td>R4</td>
<td>309.83936</td>
<td>309.95900</td>
</tr>
<tr>
<td>d</td>
<td>-0.00014392</td>
<td>-0.00014386</td>
</tr>
</tbody>
</table>

Of course the model could just as easily have been made with many more segments resulting in even greater precision.
If you solve the same model with different loading cases and save them, you can use *File/Load result files* on the load diagram dialog and display them all at once on the load diagram.

**BEAMELASFND.FEM (BEAM ON AN ELASTIC FOUNDATION)**

**DESCRIPTION**

These notes are only a brief outline. This example is outlined in much more detail in exercise 19 in Help "Getting started" along with several alternate solutions and comparisons with classical analysis.
DESCRIPTION

Concrete beam 240 inch long, 8 inch wide x 24 inch deep sitting on soil.

The soil vertical compression stiffness (spring rate per sq inch) is given as 2000 pound per inch per square inch.

The beam supports its own dead weight of 16.704 pound/inch and 500 pounds per inch external distributed load and also a load of 100000 pound load concentrated at the center representing a column load imparted to the foundation beam.

MODELING

Library properties 24 x 8 rectangular beam from the Basic Shapes module

Beam stiffness properties

\[
E = 3000000 \\
G = 1300000 \\
A = 144 \\
IY = 432 \\
IZ = 6912 \\
J = 1455.634
\]

Stress properties

\[
SY = 144 \\
SZ = 576 \\
ST = 250.4348 \\
AYF = 1 \\
AZF = 1
\]

The beam is modeled with nodes spaced every 6 inches. (e.g. you can make it in the Quick modeler with 0, 0, 0 and 240, 0, 0 and with 40 sections).

LOADS

The loads are set up and combined using the discrete and global loading dialogs.

Select only the center node where the concentrated 100000 pound load is to be applied. Use Edit/Loads from the main menu bar or Set loads from the pop-up menu. Or, if you are in the Nodal Editor, you can press the Load button on the Node editor panel.

In the dialog box, add -100000 pounds for the FY value. Leave all else at zero. Check Cartesian and Replace existing.

Press OK and the 100000 pound downward acting discrete load is applied to node 21.

Turn off the node circles so that you can select elements easily with the frame selector.

Select all of the elements in the beam. Right-click on the screen and choose Set loads.

On the dialog, open the Dead weight tab, check Y, Negative, and Merge with existing.

Enter the weight per unit length (16.704) in the intensity field, and leave the Use compensation item unchecked.

Click on the Apply dead weight button. This calculates the contribution of the dead weight to each node and merges it with the existing 100000 pound load already on the model.

Stay in the Global loading dialog and open the Projected load tab, press Y, Negative, and check Merge with existing.

Enter the force per unit length (500) in the intensity box, and leave the Use compensation item unchecked.
Click on the Apply projected load button and the projected 500 pound/inch load is merged with the existing load conditions.

The total down load (FY) should now be -22409 pounds.

Press OK to leave the Global loading dialog.

REACTIONS

The foundation spring restraints are set up by selecting all the elements (and only elements) and using Set bounds. In the elastic foundation dialog, you can choose the Normal option with a value of 3840000. This is the stiffness value for the whole beam (2000 x 8 x 240) normal to its surface. The application results in 48000 applied to the end nodes and 96000 to all others.

You can select the end nodes and manually set X and Z and Xrot as fixed in order to stabilize the model.

Finally, use Tools/Set one-way restraints and set the Y direction of restraint to be active only with positive reactions. This allows liftoff should it tend to occur.

SOLVE AND EXAMINE RESULTS

The displacement at the center is -0.09665 inches downward. A plot of displacement is possible by selecting all the nodes and using Results/Displacement diagram.

The displacement on the ends of the beam is -0.01561 inches downward.

This indicates no liftoff when all loads are applied.

If you remove the distributed load and apply only the discrete load then the displacement at the center is -0.06645 and the displacement on the ends is +0.04086 upward. Furthermore, the reaction forces there are essentially zero indicating no contact with the soil.

A classical calculation for the single discrete load (which ignores lift up on the ends) gives -0.06435.
Beam bending stress is shown by selecting the elements and using *Results/Load diagram* and toggling through the plot options.

![Beam on an elastic foundation - Help exercise 19](image)

**COMPONENT BENDING STRESS**

Max  -1690  Min  -150

**BEAMWEBCOMBO.FEM**

This type of beam could be a component that might form part of a larger model of integrated beams and plates that one intends to construct.

It is advisable to build separate components like this and study comparisons to make decisions on the modeling approach, especially when beams are to be integrated into plate element models. Beam webs are especially subject to excess stiffness if not modeled with a sufficient discretization across the depth and beams are difficult to fit into enveloping geometry that might be required by a combined beam and plate type model.

**GEOMETRY**

Use a 1.5 x 1.5 x 0.25 x 0.25 T-beam for the caps and a 0.125 inch thick plate for the web. The beam is 6 inches deep and the web is 0.125 inches thick.

You can solve this full section for the properties needed for a slender beam analysis by classical methods.

**PROPERTIES**

We used CADRE Profiler to generate the total section properties but you can easily solve for them by basic section property methods.

Output from CADRE Profile follows:
Area = 2.062500
Izp = 10.655117 (principal axis)
Iyp = 0.160936 (principal axis)
Extreme fiber distances from centroid:
CY1 = 3.000000
CY2 = -3.000000
CZ1 = 0.687500
CZ2 = -0.812500
Bending moduli (Ix/Cy & Iy/Cx):
SZ1 = Iz/Cy1 = 3.551706
SZ2 = Iz/Cy2 = 3.551706
SY1 = Iy/Cx1 = 0.234089
SY2 = Iy/Cx2 = 0.198075

The basic T beam used for the caps would have these properties (CADRE Pro Basic Shapes module).

T-section 1.5 x 1.5 x 0.25 x 0.25
Area = 0.6875
Iy (Inertia) = 7.19401E-02
Iz (Inertia) = 1.38524E-01
J (Torsion coefficient) = 1.4523E-02
Ctr to edge on Y = 4.65909E-01
Ctr to edge on Z = 7.5E-01
Sy = 9.59201E-02
Sz = 2.9732E-01

It is difficult to model the full section as shown above and meet both the 6 inch depth dimension and also the stiffness and stress properties. If the beam is to fit into another model the geometry may have to be retained as well as the properties.

Often, one can find an equivalent section more amenable to idealization as a finite element model meeting all of the requirements.

Here is one that has the web extended to the surface and the caps cutout to compensate for the added web. This would have exactly the same properties (assuming the same material elastic modulus) but the components now span or extend to the full 6 inch depth which would be essential if this beam was part of a larger component or model.
Properties of a single T section modified to allow the web to extend to the surface

![Diagram of T section]

CADRE Profiler output follows:

- Area = 0.656250
- Inertia about centroid:
  - \(I_{zp} = 0.134932\)
  - \(I_{yp} = 0.070682\)
- Extreme fiber distances from centroid:
  - \(C_y1 = 0.482143\)
  - \(C_y2 = -1.017857\)
- Bending moduli (\(I_x/C_y\) & \(I_y/C_x\)):
  - \(S_{z1} = I_z/C_y1 = 0.279719\)
  - \(S_{z2} = I_z/C_y2 = 0.132498\)

We developed the above using the CADRE Profiler application but they could very easily be determined by simple section property calculations.

This modified cap would be defined as an eccentric beam; it is defined in the element library with the name "T-ecc-mod".

In all cases, the distance from the centroid to the top of the flange is the key off-center distance so the values determined and used for stress (\(C_y\) and \(S_z\)) are the Near edge values.

THE MODEL

The model is constructed with a web that is 6 x 96 inches and divided by 8 sections in depth and 96 in length. The discretization in depth is the greatest contribution to accuracy. Too few sections in depth will result in an overly stiff beam. Using 8 sections will still be a little stiff by about 6% but the web is only a small part of the overall beam rigidity (caps dominate!) so the overall accuracy will be much better and within acceptability. All FEA models are approximations and compromises!

The spar caps are attached all along the top and bottom of the web at every node. We solved for several cases so that one may compare the results. The first solution below is the simple classical beam analysis. This is the usual one for comparison, but keep in mind that the classical slender beam is not an 'exact' solution and full of approximate assumptions as well.

The beam is fixed on the left end, and loaded with 600 pounds (total) on the right end.

ALTERNATIVE CAPS FOR COMPARISON

1) T-std is the full T beam properties 1.5 x 1.5 x 0.25 x 0.25 considered as a single line beam concentrated on the top and bottom of the web.

2) T-ecc is the same T beam again but defined as an eccentric beam with off center value \(D_y = -0.465909\) (per \(C_y1\) above of T-std)

3) T-ecc-mod is the modified T beam with the cutout with off center value \(D_y = -0.482143\) (per \(C_y1\) above of T-ecc-mod)

Use near edge for \(S_z = 0.134932/0.482143 = 0.279859\)

SOLUTIONS

Classical slender beam analysis

\[y = \frac{PL^2}{(3EI)} = 1.66\text{ inches}\]
Sn = M/Sz = (600)(96)/3.551706 = 16217 psi at the root

1) Using the basic T-std as a 'standard beam' without accounting for neither the cutout nor the off center centroid of the caps.
   \[ y = -1.1939 \]
   \[ Sn = 14548 \text{ psi} \]

2) Using the basic T-ecc as an eccentric beam but without the cutout compensation
   \[ y = -1.5610 \]
   \[ Sn = 16319 \text{ psi} \]

3) Finally, using the T-ecc-mod cap as an eccentric beam including the compensating cutout
   \[ y = -1.6332 \text{ in} \]
   \[ Sn = 17118 \text{ psi} \]

Difference in stress from the classical analysis can partially be attributed to the 2D approach to restraint at the edge which causes some increased wrenching in the last inch or so of the beam. Such anomalies are present in the real structure as well so which is the "correct" stress is a matter of for debate.

WEB ALONE

Finally as a check of the web acting alone, set the Caps to a line element and solve in CADRE Pro.
   \[ y = -7.3687 \text{ in} \]
   \[ Sx = 76710 \text{ psi} \]

Compare this to the displacement of a slender beam classical analysis of just the web acting alone
   \[ I = 2.25; c = 3.0 \]
   \[ Sz = 2.25/3.0 = 0.75 \]
   \[ y = PL^3/(3EI) = -7.8643 \text{ in} \]
   \[ Sn = 600 \times 96/0.75 = 78600 \text{ psi} \]

The depth is a little low in comparison but, when combined in the full component with caps, the web is only a small part of the structural stiffness so the contribution of the web to the inaccuracy in displacement would also be smaller and perhaps not so significant.

A better way to compare with classical analysis would be to compare at a station slightly removed from the restraint by at least one element (i.e. calculating stresses 95 inches inward from the load rather than 96).

CONCLUSION

One could improve on the accuracy by increased divisions on the depth but you may find that the full model in which you want to build this beam or spar would have to be very dense and perhaps unsolvable. You may even have to accept a reduced count on the depth and a little more inaccuracy to achieve a result. Using only 6 divisions on depth doesn't change the inaccuracy in displacement very much and is still very good in stress. In any case, stress results are conservative.

These are the kinds of studies, comparisons, and compromises, that one should do to help make decisions on the modeling scheme before developing the full model.

Structural models of airplane wings with tapered and twisted spars, stringers and ribs have been constructed with very good structural analysis results. The quality of the final model will depend on the decisions made early with the simple component studies as demonstrated in this example.
BOLTHOLE.FEM (Bolt in a hole model)

BOLT IN HOLE MODEL

This model illustrates:
1) The use of *Pinned compression beams* to represent contact between elements
2) The use of the *Hole template*
3) The use of the *Spheric template* to make flat disks

CONSTRUCTION

Create two independent models from templates

Use *Utilities/Hole model*

- Upper right corner: X = 1, Y = 1
- RADIUS = 0.28125 (9/16 diameter)
- Divisions:
  - Horizontal 3
  - Vertical 3
  - Radial 5
- Log style
- Check: Evenly distribute rim nodes
  - Note: There are 24 rim nodes matching the Spheric above!

In the Element Editor mode, select and assign the group name of “HOLE” to this model.

*OK* and then save this model for now.

Use *Utilities/Quick modeler* and open the ‘Spheric’ template.

- Height: 0.0
- Radius: 0.25
- Rings: 6
- Sectors: 6
- Iso-adjusts: 2
  - Supports: 24 (matching the hole rim nodes below)

In the Element Editor mode, select and assign the group name of “BOLT” to this model.
OK and Save this spheric model for now.

BUILD THE COMBINED MODEL

Start with the Spheric model.

Select the center node, the use Select/Nodes/by group and select the “EDGE” nodes.

Use Copy nodes relative. Choose the Radial relative to first selected node option, Set Y = +0.03125, and check Add connecting beams to the copies.

Press OK and the bearing connectors beams are all installed at once.
Select the upper node, the center node, and the node on the right (in order).

Right-click and use **Merge model**

Choose the previously saved **Hole** file from the file dialog file list.

Choose corresponding nodes on the hole model (upper, middle, right).

Use **Continue** to merge the models.

The BOLT is now merged into the HOLE along with connecting beams that will become one-way pinned compression beams to represent the bearing surface.

Plate elements have **no rotary stiffness** about a node normal to the plate so you can’t restrain a plate from in-plane rotation at a node unless there is some other element to transfer load (or in this case restraint) to the plate elements.
Add a two-segment stabilizer beam across, and connected to, the center node.

All the beams must have an orientation node which is not collinear with any of them. Select the center node and use *Copy nodes relative* (uncheck the option to add connecting beams!). Set Z = 0.5 above the center node out of the plane of the model.

Select all the connecting studs and stabilizer. Assign the new out-of-plane node to them for orientation. You can then select and hide that out-of-plane node so that the bolt center node is more easily accessed.

That completes the geometric aspects of the model. You can use the *Bandwidth Manager* under *Tools* at this point to arrange the model into a more efficient form.

**ELEMENT PROPERTIES**

1) **STUD**: interconnect beams are somewhat arbitrary but make as simple rods about the same diameter as plate thickness. Set the type to *Compression beam (pinned)* and set pin Y as zero for both Axis and Origin and set pin Z as 1.0 for both axis and origin.

![Element Library]

2) **STABILIZER**: The stabilizer bar is even more arbitrary as long as bending stiffness isn't zero. Make same as interconnect beams in size but make it a *Standard beam* type.

3) **STEEL PLATE 0.1**: Kirchhoff steel plate at T = 0.1

4) **STEEL BOLT 0.5 X 0.25**: Kirchhoff steel plate at T = 0.25 (Not very important since the focus is on the flange hole stress).

Enter these in the element library and then assign them to the appropriate parts of the model.

**LOADS**

- FX = 350
- FY = 200
- RESULTANT = 403.11 LB
RESTRAINTS

Center node: restrain Xrot, Yrot, Zrot only, all else free (slide but no rotations)
Hole surround edge nodes: restrain in X, Y, and Z (i.e. pinned)

SOLUTION

Principal stress S1 about 8099 psi

The above representation, using an actual disk for the bolt, is really not necessary. You can build a simpler model by not using the disk and just drawing radial connector beams from the center of hole to the rim. The results are essentially the same.

Then again, just performing manual hand calculations for this type of analysis is even simpler than building any finite element model and more consistent with code specifications.

**BOXBEAM.FEM (BUILT-UP BOX BEAM)**

This is the model examined in Exercise 1 of the getting started tutorial mainly as practice in viewing and manipulating models.

The structure is a built-up box beam consisting of a simple frame and sheet construction. The round ‘apparent’ opening in the loaded end is structurally closed and the same as the visible portion of the end but it has been set as invisible so that one can see inside.
The elements of the "opening" are just identified as a separate definition in the Library (with the same properties) except set to 'hidden'. One could also reduce the opacity of that region, or any other part as well to achieve a view inside. In a real structure, it could be a structurally covered inspection hole.

There is a stored custom viewpoint matching the position illustrated in Exercise 1. You can use Ctrl-U and pick it from the list to orient the model to the exact orientation shown in the exercise. Stored viewpoints are useful when you need to send the model to someone else and ask them to look at it from a particular point of view.

COMPARISON TO CLASSICAL METHODS

It is often useful to compare finite element methods with older classical methods. This model represents the problem description for an exercise given in the original first edition of Peery, Aircraft Structures.

ANGLE OF TWIST: The Peery 'classical' solution initially assumes no warping of the cross section anywhere along the span and the final calculated twist given by classical analysis is 0.02 radians. The exercise goes on to estimate the error due to warping to be 0.00362 radians; giving a total of 0.0236 radian for the final answer. This finite element solution gives a value of 0.02357 radians for the total twist of the end section.

DISPLACEMENT: The deflection at the loaded point is calculated in Peery by classical methods to be 5.96 inches. This finite element solution gives a value of 5.955 inches.

This sample model is also referenced in the CADRE Pro user manual showing how to apply the taper and twist features to build a box beam that is initially untapered and/or untwisted. Another example in the CADRE Pro user manual uses this model and shows how to replicate the end wall rib bulkhead at stations along the interior.

(Ref. Peery McGraw Hill, 1950, Chapter 16, Page 443)

**BRIDGE.FEM (2D TRUSS FOR STUDYING STABILITY AND BUCKLING)**

This example represents a simple 2D truss using the inch pound system of units. It is set up to illustrate buckling and instability using the Advanced analysis feature. The diagonal struts have been purposely designed with section properties which will allow buckling at loads less than the full applied load.

This sample model illustrates:
- Stability and buckling
- User defined element names
- Reference element lengths
RESTRAINTS

The node points along the upper chord are restrained (only in translation) in the 3rd (Z) dimension direction so as not to allow movement in that dimension in the buckling check. It makes no difference for this model but lateral in/out buckling of the upper chord in the Z dimension could be an issue on some models. It just depends on whether you are trying to simulate a 2D analysis of the truss or not since all models are really 3D unless forced to respond as 2D.

The two side supports (1 and 11) are restrained vertically and only one is restrained in the lateral (X) direction. This is sufficient to prevent rigid body lateral motion of the structure but still allow loads to be transmitted into the lower chord from the side support and prevents elastic binding of any elements.

The diagonal members are pinned at both ends. You can check the pins by selecting one of the diagonals, right-click, and choose Show beam pins.

You can always set pins for a standard or pinned beam by selecting it and using Tools/ Set beam pin factors. The pin setting dialog allows you to set and reset beam pins and watch the results directly on the model.

LOADS

There are four load points on the lower chord.

Always set the loads view to "Local" loads with the "L" tool bar button (not Global with "G" which is rarely used and usually for troubleshooting balance at a joint).

SOLVING

After solving you can check individual internal loads by highlighting the element with mouse click and then use the eye view tool. The Fx value is the axial end-load. You can continue to check other elements without closing the eye view tool.
Use **Solve/Advanced** with only 1 increment to show that member P*4*5*6 (or "e101") as unstable.

This model has special user defined names (e.g. e101) which can optionally be displayed and optionally be shown in lieu of the default system names (P*4*5*6).

Use **Solve/Standard** to show that this member would have an elastic load of 606 pounds if allowed to continue without considering buckling.

The Euler buckling load from classical analysis for this pinned strut (283 inches long) is:

\[ P = \frac{\pi^2 EI}{L^2} \]

\[ P = 589 \text{ pounds.} \]

Using **Solve/Advanced** with 10 increments - buckling between 90 - 100% >> 0.9 \times 608 = 545 pounds

Using **Solve/Advanced** with 50 increments - buckling between 96 - 98% >> 0.96 \times 608 = 582 pounds

Use **Solve/Standard** and examine **Results/Axial Loads** and see that there is another member (e.g. e106 on the opposite end) that is unstable. The one identified during stability checking is just the first one found by the buckling algorithm at which point the solution ceased. Unstable values are shown in Red font.

**REFERENCE LENGTHS**

The **Results/Axial load** output is based on the element length OR the assigned reference length if one is assigned.

This model has already been assigned 'reference lengths' to all elements which are equal to the actual member lengths. This is accomplished (always before subdivision!) by selecting the elements in the Element Editor mode. Right-click and use **Set lengths**. Set to the current member length. Then, one can subdivide the model as desired and the **Axial load** output under **Results** will always be based on the member reference length even though each segment of member is shorter.

For example: Select all elements in the model. Right-click and use **Divide**. Divide all members by 5.

Now **Solve/Standard** and use **Results/Axial loads**. Note that the same members are unstable including every individual segment of those members.

You can also use **Solve/Advanced** and show that the stability flagging results at the same levels of load.

The reference length assignment can also be useful if one wishes to assign and "effective length" to columns before stability solutions.
CABLEBRIDGE.FEM (3D cable-stayed bridge)

This model illustrates:
- pivot beam intersections
- tension cable elements
- rigging loads (preloads)
- Options for dealing with mid-panel loads

DESCRIPTION
Span = 850 feet (10200 inch)
Width = 75 feet (900 inch)
Tower height = 300 feet above ground restraint; 200 feet above bridge bed
Tower cant = 2.386 degrees
Towers 48 inch square x 2.5 inch steel
Bed is 15 inches concrete

CABLE STAYS
These are modeled as tension-only elements with zero bending and torsion properties.
The special size is 5.0 inches diameter. The wire rope configuration is 6 X 37.
Import the wire rope properties from the USA Steel section property database.

The wire rope elements are set as Tension beams.

BRIDGE TO TOWER CONNECTION

These connections are free pivot supports installed using *Special structures/Insert pivot beam*.

The horizontal member passes continuously across the pivot yet is free to pivot (see-saw) on the tower node.

RESTRAINTS

One advantage of cable-stayed bridges over suspension bridges is that a large anchor lateral system is not needed at each end of the bridge. In this model, bridge ends are allowed free to slide (X restraint is free) and after solution the movement is only about a quarter of an inch (compression in the girders) which is insignificant relative to the 850 foot length.

The tower bases are conservatively pinned in this model. One could provide whatever ground restraint is actually present such (e. g. ground springs based on the size of the footings). For the vertical loading condition considered here, it makes little difference but could be important for lateral (wind) loads.

Lateral X restraints are set up at the point of symmetry in the middle of the bridge. After solution, the reaction is essentially zero so they do nothing more than prevent rigid lateral freedom to allow for a stable solution.

LOADS

Dead weight = 15.34E6 lb (including towers, structure, and cable)
Additional load = 50 lb/foot on 850 feet span = 510000 lb
Actual surface area is 9.126E6 (reduced by small area at tower intersection)
Surface intensity = 510000/9.126E6 = 0.0558843 (applied to plate elements)
Total load = 15.85E6 lb
Since the bed surface material is not represented in enough detail in this model for actual stress analysis, the loads are shifted off of mid-panel nodes and onto frame supported nodes using the special loading feature *Shift panel loads to frame*.

**PRELOADS**

The cable stay elements are selected and assigned a preload tension value of 40000 pounds. Use *Set preload* from the pop-up menu. A value of -4000 pounds of preload is a tension preload.

![Assign preloads & temperature to selected elements]

**SOLVE**

After solution check Results/Beam stress maximum

**MAXIMUM/MINIMUM NORMAL STRESS, SN1, OR SN2 - MAXIMUM OF EITHER END**

<table>
<thead>
<tr>
<th>Group name</th>
<th>Max/Min</th>
<th>Max/Min</th>
<th>Max/Min</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lateral beam</td>
<td>39549</td>
<td>-39298</td>
<td></td>
</tr>
<tr>
<td>Main girder</td>
<td>25178</td>
<td>-37020</td>
<td></td>
</tr>
<tr>
<td>Stays</td>
<td>13990</td>
<td>4569</td>
<td></td>
</tr>
<tr>
<td>Tower</td>
<td>-2879</td>
<td>-11726</td>
<td></td>
</tr>
</tbody>
</table>

**CIRFOUND.FEM**

These two examples, Part I and Part II, serve as good basic benchmark checks for CADRE Pro in that they are well known exercises with well known results.

**PART I**

This is an example of a plate on an elastic foundation from Roark *Formulas for stress and strain*, 5th edition, section 10.6.

It is a steel disk simply supported at the edge. It sits on a 20 pound/inch per square inch elastic foundation with a 3 psi applied pressure.

The disk is 20 inch in diameter and 0.2 inch thick.

**CONSTRUCTION**

Use the *Quick modeler* and choose the 'Spheric' template. With height set to zero, the *Spheric* template is actually a flat disk template.
Set: XY Plane, Plates only, Center at (0, 0, 0), Radius = 10, Height = 0, Rings = 8, Sectors = 8, No. Iso adjusted = 2

ELEMENT DEFINITION
Enter (Add) the element library definition and set the type as Kirchhoff plate.
Set properties E = 3E7, V (Poisson) = 0.285. T (thickness) = 0.2
Give it a name such as ‘Steel plate’

ASSIGN THE PROPERTIES
Go to the Element Editor mode
Select the whole model with the frame selector.
Use the ‘Properties’ button on the editor panel to assign the single Library property to all the plate triangular elements in the model.

LOADING
Method 1 (Apply Library F/A value to the whole model using library parameters)
- Set F/A normal pressure parameter of 3.0 psi in the Element Library for the defined plate element
- Use Edit/Loads or Set loads from any pop-up menu without selecting any nodes or elements.
- Use the Projected tab in the +Z direction (or use the Normal tab positive direction).
The 3 psi value for the plates assigned from the element library will be distributed over all the nodes.
Method 2 Recommended (Apply user supplied value to selected elements)
Do not select any nodes but;
- Select the entire disk with Select/Elements/All or drag the frame selector over all elements.
- Use Edit/Loads from the main menu, or Set loads from any pop-up menu.
- Enter 3.0 in the intensity field on the loading dialog.
- Use the Projected tab in the +Z direction (or use the Normal tab positive direction).
The 3 psi value will be distributed over all the nodes of selected elements (all plate elements in this case).
The total load will be 939.79 lb.

ELASTIC FOUNDATION RESTRAINTS
The nodes are to be defined as spring restraints to represent the elastic foundation.
Some special convenience options are available for elastic foundations.
Calculate the foundation gross vertical stiffness:
\[ \text{Ktotal} = \text{Area} \times 20 \text{ lb/in per sq in} = 6283.19 \text{ lb/in} \]

Ensure that no nodes are selected!

Select all of the plate elements (i.e. turn off the node circles and drag the selection frame over the whole model or use \textit{Select/Elements/All}.)

Right-click and pick \textit{Set bounds}.

Check the 'Normal' Option and enter the value 6283.19 in the normal total value field.

Press \textbf{OK} and the total stiffness will be distributed over all the nodes of the selected elements proportional to their area (including the edge nodes).

\textbf{OTHER RESTRAINTS}

For convenience, the edge nodes have a group name 'edge'.

Under main menu \textbf{Select/Nodes/By Group} select the "EDGE" nodes.

\textit{Use Edit/Bounds} or from any pop-up menu use \textit{Set bounds}.

Use the \textit{Pin} button to set the rim nodes to the pinned state as required by the problem statement.

Note: Always set the elastic foundation first, then the discrete restraints afterward!

\textbf{SOLUTION}

Solve standard.

\textbf{CADRE Pro results:}

- S2 principal (centroid) = 6467
- S2 principal (mid side) = 6479
- S2 principal (vertex) = 6513
- Zmax = 0.0634 at center node

Classical analysis provided by Roark results in: Smax = 6504 psi; Zmax = 0.0637 inch
While solved you can select a path of nodes from left to right across the disk, then use Results/Displacement diagram to plot the displacement. If you select by frame rather than one-by-on in order, then you would use Custom re-order and with increasing X (i.e. Select/Nodes/Re-order selection or (Ctrl J).

Spheric
Z DISPLACEMENT Max = 0.0634 Min = 0.0000

PART II
This is another exercise from Roark "Formulas for Stress and Strain", 5th Ed., section 10.2 that can be solved using the same model already developed above. The essential difference is that there is no elastic foundation.

The situation is a pressure chamber resisted by a steel disk. Initially the steel disk simply supported at the edge. It has 3 psi applied pressure from the chamber. The disk is 20 inch in diameter and 0.2 inch thick.

Open the cirfound.fem model and use Select/Nodes/By group.
Select all groups except "EDGE".
Use Set bounds and use the Unassign button to remove restraints from the selected nodes. That removes all restraints (i.e. elastic foundation) except at the pinned supports on the rim.

SOLUTION
Roark classical results: Smax = 9240 center; Yc = 0.0883 inch
CADRE results (high vertex S1): Smax = 9305 center; Yc = 0.0883 inch

ALTERNATE BOUNDARY CONFIGURATION
Alternate boundary exercise: Fix the rim nodes in all degrees of freedom and resolve.
Roark classical results: Smax = 3614 center; Smin= -5625 edge; Yc = 0.0215 inch
CADRE results (high vertex S1): Smax = 3709 center; Smin = -5777 edge; Yc = 0.0216 inch
CODEALUMINUM.FEM (Checking against the aluminum design code)

This sample model demonstrates the use of the Aluminum code checking feature for combined axial and bending loads. This is a 50 inch long aluminum (6061-T6) rectangular beam 4 x 2 x 0.125 wall.

The 50 inch long idealistic beam model was created in the Quick modeler with the Beam template using 10 segments.

ELEMENT LIBRARY

Go to the element library and set up the Library element properties with 4 X 2 X 0.125 aluminum beam section. You can import it directly to the library from the "Alum USA" database. Then, revise the E modulus value to be precisely 10.1E6 to match the specific aluminum product for this exercise. Select and assign the property to the entire beam.

REFERENCE LENGTH

Since column buckling of the full member is one consideration in the analysis, you MUST set up reference lengths so that all segments of every member are treated as if they are full length of the member in which they are a part. If there were multiple members, you would set the reference lengths of all members at once before subdividing. There is only one member in this model and it has 10 segments.

In the Element Editor mode, select all 10 of the beam segments, right-click and choose Set lengths from the pop-up menu. Enter 50 for the reference length of all selected segments.

LOADS

SELECT NODE 6 AND SET ITS LOAD, THEN NODE 11 AND SET IT.

Node 6 at the center Fx = 0; Fy=-600; Fz = -50
Node 11 at the right end Fx = -1000, Fy=0; Fz=0
RESTRAINTS

Node 1: XXXX00  (fix X, Y, Z, Xrot) (free Yrot, Zrot)
Node11: 0XX000   (fix Y, Z) (free X, Xrot, Yrot, Zrot)

STRENGTH ALLOWABLES

Strength allowable data for 6061-T6 (psi) from the design code.

\[
\begin{align*}
E &= 10.1E6 \\
F_{tu} &= 38000 \text{ psi} \\
F_{ty} &= 35000 \text{ psi} \\
\nu &= 1.95 \\
\nu_y &= 1.65 \\
F_{cy} &= 35000 \text{ psi} \\
F_t &= \min\left(\frac{F_{ty}}{\nu_y}, \frac{F_{tu}}{\nu}\right) = 19487.2 \text{ psi} \\
F_{by} &= 19487.2 \\
F_{bz} &= 19487.2
\end{align*}
\]

CONFIGURE TO USE THE ALUMINUM CODE CHECKING FEATURE

Use Options/Settings on the beam tab and check Use code check

Choose the "Aluminum" code option.

This code checking system is adapted from the Aluminum Association specification (ASD version)

Only the combined bending/compression or tension on gross compact sections is treated by this check.

Solve the model then use Results/Code results/Aluminum beam stress

In the dialog, set up just $F_{tu} = 38000$, $F_{ty} = 35000$, $\nu = 1.95$, $\nu_y = 1.65$. 
Then use the **Set** button to fill in the remainder of the dialog. This sets Fby and Fbz to 19487.18. Leave Ky, Kz, Cmy, Cmz all as 1.0 for this model with pinned ends.

![Aluminum stress ratios](image)

Press **OK**.

In the next dialog, check ‘Maximum of either end’.

You can use the **Custom** button and uncheck the items for Principal, Normal, and von Mises stress, leaving only the Components and Length columns to be displayed in the table. (Note: You could make a custom selection the default from **Options/Settings** on the Results tab).

![Beam stress location](image)

Press **OK**.

The solution option outputs component stresses and stress ratios relative to several different requirements for every element (segment of a member in this case) in the model. The largest ratio of any requirement is in the column labeled ‘R’. The summary row gives the maximum of all elements. Scroll the table to the bottom and read_

Rmax = 0.346,

at the bottom of the R column.
Also note the Reference is (C) 4.1.1-3 which is the specific paragraph of the requirement deriving the critical stress ratio.

<table>
<thead>
<tr>
<th>R</th>
<th>Ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.113</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.172</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.230</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.288</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.346</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.346</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.288</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.230</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.172</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.113</td>
<td>(C)4.1.1-3</td>
</tr>
<tr>
<td>0.346</td>
<td>(C)4.1.1-3</td>
</tr>
</tbody>
</table>

You can repeat this operation using the *Aluminum beam stress maximum* option which will present only the maximum and minimum stresses (and referenced location) sorted by group or type rather than data for every segment. The initial dialog allows one to choose additional options for the display using check boxes.

![Screenshot of CADRE PRO sample model files](image)

There is only one property type for this model and no group names so only one line of data will be provided under the headings in the actual output table for this model (here it is split into 3 lines so that all the columns will fit).

<table>
<thead>
<tr>
<th>ALUMINUM STRESS RATIOS - Maximum of either end</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property name</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>RECT TUBE 4.0 x 2.0 x 0.125</td>
</tr>
<tr>
<td>Property name</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>RECT TUBE 4.0 x 2.0 x 0.125</td>
</tr>
<tr>
<td>Property name</td>
</tr>
<tr>
<td>----------------</td>
</tr>
<tr>
<td>RECT TUBE 4.0 x 2.0 x 0.125</td>
</tr>
</tbody>
</table>
Even if the model had thousands of elements, but only a few dozen different types or group names, a table like this would be the preferred output since it gives the critical stresses and stress ratio for every different type or group.

MANUAL CALCULATION FOR COMPARISON

\[ Bc = \frac{Fcy(1+Fcy/2250000)^{0.5}}{2} = 39365 \]
\[ Dc = \frac{Bc}{10(Bc/E)^{0.5}} = 245.76 \]
\[ Cc = 0.41(Bc/Dc) = 65.673 \]
\[ S1 = \frac{Bc - (Nu/Ny)(Fcy)}{Dc} = -8.13 = 0 \]
\[ S2 = Cc = 65.673 \]

YAXIS:

\[ Ry = \sqrt{\frac{Iy}{A}} = \sqrt{\frac{(0.99186/1.4375)^{0.5}}{0.83065}} = 0.83065 \]
\[ KL/Ry = (1.0)(50)/0.83065 = 60.19338 \text{ (between S1 and S2)} \]
\[ Fay = \frac{(1/Nu)(Bc-Dc(kl/Ry))}{(kl/Ry)^2} = 12601.13 \text{ psi (column limit)} \]

ZAXIS:

\[ Rz = \sqrt{\frac{Iz}{A}} = \sqrt{\frac{(2.97624/1.4375)^{0.5}}{1.43890}} = 1.43890 \]
\[ KL/Rz = (1.0)(50)/1.43890 = 34.7488 \text{ (between S1 and S2)} \]
\[ Fa = Min(Fay, Faz) = 12601.13 \]

CALCULATED STRESSES AT NODE 6 FROM CADRE PRO:

\[ Sx = -696 \text{ psi (axial stress)} \]
\[ Sby = 630 \text{ psi} \]
\[ Sbz = 5040 \text{ psi} \]

\[ Dpby = Fby*(1-Abs(Sx/Fey)) = 18526.35 \]
\[ Dpby = Fbz*(1-Abs(Sx/Fez)) = 19166.99 \]

\[ R(0) = Abs(Sx / Fa) + Abs(Sby / Dpby) + Abs(Sbz / Dpbz) \]
\[ R(0) = 0.03281 + 0.03233 + 0.25863 = 0.3238 \]

Aluminum Association ASD Section 4; Equation 4.1.1-2

\[ Fa0 = Fcy/Ny = 21212.12 \text{ psi} \]
\[ R(1) = Abs(Sx/Fa0) + Abs(Sby/Fby) + Abs(Sbz/Fbz) \]
\[ R(1) = 0.03281 + 0.03233 + 0.25863 = 0.3238 \]

Aluminum Association ASD Section 4; Equation 4.1.1-3

\[ R(2) = Abs(Sx/Fa) + Abs(Sby/Fby) + Abs(Sbz/Fbz) \]
\[ R(2) = 0.05515 + 0.03233 + 0.25863 = 0.3462 \]

Since Sx/Fa < 0.15 one may use 4.1.1-3 in lieu of either 4.1.1-1 or 4.1.1-2

\[ R = 0.3461 \text{ is chosen for Rmax (ignoring the 0.3522 value!).} \]

CODESTEEL.FEM (Checking against the steel design code)

The model is a W10x15 steel section 100 inches long pinned ends. Steel type: Fy = 36ksi
CONSTRUCTION

This model is made in the Quick modeler at 100 inches long and with 10 segments.

Set up the element library by importing the W10x15 section from the Steel USA database.

RERAINTS

Node 1: XXXX00 (fix X, Y, Z, Xrot) (free Yrot, Zrot)
Node11: 0XX000 (fix Y, Z) (free X, Xrot, Yrot, Zrot)

REFERENCE LENGTH

Since column buckling of the full member is one consideration in the analysis, you MUST set up reference lengths so that all segments of every member are treated as if they are full length of the member in which they are a part. If there were multiple members, you would set the reference lengths of all members in the model before subdividing. There is only one member in this model and it has 10 segments.

In the Element Editor mode, select all 10 of the beam segments, right-click and choose Set lengths from the pop-up menu. Enter 100 for the reference length of all selected segments.

Use OK to set the reference lengths of the elements to 100 inches.

CONFIGURE CADRE PRO FOR THE STEEL CODE

Use Options/Settings and open the ‘Beam tab’ and check Use code check.

Choose the "Steel" code option.

The code check feature allows one to use the 1989 and earlier methods, or the 2010 and later methods. The 1989 code is more conservative in some ways than the 2010 and later codes so many users prefer to use those traditional methods for initial sizing of the structure.

CODE 1989

This method is from Chapter H of the 1989 code and is limited to compact sections with doubly and singly symmetrical members. Only the combined bending/compression or tension on the gross section is treated by this check (torsion and shear are ignored!). Since there are no other checks than simple bending combined with compression the older 1989 code is more likely to be within compliance to the current code when considering all the additional checks with varied section types that may be included in the members chosen for evaluation under an existing code.

Load the model with the compression case, or the tension case as described below.
COMPRESSION MODEL 1989

LOADS

Node 6: Set loads; Fx = 0, Fy = -3500, Fz = -300
Node 11: Set loads; Fx = -6500

After solving the model, select the segment to the left of center. Notice that its axis node is at node 6 which is the very center of the beam.

Use *Results/Code result/Steel beam stress*.

In the allowable strength dialog, enter the strength and configuration properties.

One way to enter the strength data is to enter $F_{tu}$, $F_{ty}$ then $N_u$ and $N_y$ (i.e. 58000, 36000, 2, 1.66667, then use the *Set* button. This will set all the other standard strength limits $F_{by}$, $F_{bz}$, and $F_t$ according to the $N_u$ and $N_y$ settings.

However, the major axis bending ($F_{bz}$ in this case) value can be increased be $0.66F_{ty}$ for compact sections without slender flanges.

$F_{tu}=58000$
$F_{ty} = 36000$
$F_{by}=21600$
$F_{bz} = 23760 = 0.66F_{ty}$
$F_t = 21600$

Set $K_y$, $K_z$, $C_{my}$, $C_{mz} = 1.0$ for this beam-column configuration.

$N_u = 2$: $N_y = 1.67$

No rigid settings for this span
In the next dialog Choose "Axis node" (right end of the selected segments)

SOLVED COMPONENT STRESSES - Axis end

**BEAM STRESSES - Axis end (SELECTED)**

<table>
<thead>
<tr>
<th>Element</th>
<th>Node</th>
<th>Fx/A</th>
<th>Fy/A*</th>
<th>Fz/A*</th>
<th>Mx/St</th>
<th>My/Sy</th>
<th>Mz/Sz</th>
</tr>
</thead>
<tbody>
<tr>
<td>S5<em>6</em>12</td>
<td>6</td>
<td>-1474</td>
<td>-397</td>
<td>34</td>
<td>0</td>
<td>5172</td>
<td>6345</td>
</tr>
</tbody>
</table>

Axial stress = -1474  
Shear Y = -397  
Shear Z = 34  
Torsion = 0  
Bend My = 5172  
Bend Mz = 6345  

Limit compression, Fa = 9802 psi

The stress ratio is:

maximum of

Equation H1-1 >> R0 = 0.701  
Equation H1-2 >> R1 = 0.575  
or

Equation H1-3 >> R2 = 0.657 (only when Sx/Fa < 0.15 and advantageous)

Therefore: Choose R = 0.701 from equation H1-1
TENSION MODEL 1989

LOAD

Node 6: Fx = 0, Fy = -3500, Fz = -300
Node 11: = +6500

Again use Results/Code results/Steel beam stress.

The stress allowable dialog should be set up as before.

In the next dialog choose the Axis node of the selected elements.

<table>
<thead>
<tr>
<th>Element</th>
<th>Node</th>
<th>Fx/A</th>
<th>Fy/A*</th>
<th>Mx/St</th>
<th>My/Sy</th>
<th>Mz/Sz</th>
</tr>
</thead>
<tbody>
<tr>
<td>S5<em>6</em>12</td>
<td>6</td>
<td>1474</td>
<td>-397</td>
<td>34</td>
<td>0</td>
<td>5172</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>6345</td>
</tr>
</tbody>
</table>

APPLIED STRESS:

Axial stress = 1474
Shear Y = -397
Shear Z = 34
Torsion = 0
Bend My = 5172
Bend Mz = 6345

Equation H2-1 >> R3 = 0.575 (Tension)

CODE 2010

The bending strength on the major strength axis (Fbz in this case) can be increased to:

\[ Fbz = Fy \cdot \frac{Z}{S} / Ny = 0.694Fy \] (see AISC 2010 Chapter F, Eq F2-1)

From the AISC manual for the W10X15 section, Z is 16 and S is 13.8 on the major axis.

\[ Fbz = Fy \cdot \frac{1}{1.67} \cdot \frac{16}{13.8} / Ny = 0.694Fy \] (comparable to the 0.66*Fy value in the 1989 code)

Ftu = 58000
Fty = 36000
Fby = 21600
Fbz = 25043 = 0.694Fty
Ft = 21600

COMPRESSION MODEL 2010

Limit compression, Fc = 9681 psi

(C)H-1a = 0.590
(C)H-1b = 0.569

Use equation (C)H-1a since axial stress divided by limit compression is greater than 0.2
R = 0.569

**TENSION MODEL 2010**

Limit tension, \( F_t = 21600 \)

\( (T)H-1a = 0.506 \)

\( (T)H-1b = 0.527 \)

Use equation (T)H-1b since axial stress divided by limit tension is less than 0.2

\( R = 0.527 \)

**CODEWOOD.FEM (Checking against the wood design code)**

This is a simple beam used in Help and the User Manual to demonstrate use of the AWC National Design Specification (NDS) check feature.

The example is taken directly from the NDS Commentary example C3.9-3.

Assuming pinned connections for purposes of illustration, a 3 ft section (between panel points) of a No. 2 Southern Pine 4 x 2, top chord of a gable end, parallel chord truss is subjected to axial compression forces of 300 lb (DL) and 600 lb (SL), plus concentrated loads of 60 lb (DL) and 120 lb (SL) applied at the center of the 3 ft span on the wide face, and a concentrated load of 120 lb (WL) at the center of the span on the narrow face. Lateral support is provided at the ends only. Check the adequacy of the member for bending and compression for all load combinations.

The 36 inch long beam model is divided into 10 sections and each segment has a reference length equal to the full beam length of 36 inches.

![Beam diagram](image)

**Loads:**

Node 6

- \( F_x = 0 \)
- \( F_y = -120 \) (wind)
- \( F_z = -180 \) (120 snow and 60 dead)

Node 11

- \( F_x = -900 \) (600 lb snow and 300 lb dead)
- \( F_y = 0 \)
- \( F_z = 0 \)

**CONFIGURE CADRE PRO FOR THE WOOD CODE**

Use **Options/Settings** and open the ‘Beam tab’ and check **Use code check**.

Choose the "Wood" code option.

When the model is solved and with the wood code set, then there is an additional output item under **Results** on the main menu bar.

Use **Results/Code result/Wood beam stress**.
In the allowable strength dialog, enter the strength and configuration properties as outlined below.

Refer to the User manual for the details of developing the various factors and coefficients from the National Design Standard for Wood Products.

Strength settings:

\[
\begin{align*}
F_c &= 1650 \\
F_{by} &= 1650 \text{ 'flat use (1.10)} \\
F_{bz} &= 1500 \\
F_t &= 650 \\
K_{ce} &= 0.3 \\
K_{be} &= 0.4384 \\
L_{ey}/L_{u} &= 0 \\
L_{ez}/L_{u} &= 1.661 \\
c &= 0.8 \\
\end{align*}
\]

Duration factor:

\[
\begin{align*}
\text{Wind} > C_d &= 1.6 \\
\text{Snow} > C_d &= 1.15 \\
\text{Dead} > C_d &= 0.9 \\
\end{align*}
\]

**SOLUTION CASE 1:**

Loaded with D+W+S with \( C_d = 1.6 \)

One uses the highest factor associated with any of the load components (D, W, or S).

<table>
<thead>
<tr>
<th>Wood stress ratios</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>File</strong></td>
</tr>
<tr>
<td>This will display each member set (group name or library property) along with the maximum NCS stress ratio. (NOTE: ( K_{rigid} = 0.50, K_{pinned} = 1.0 ))</td>
</tr>
<tr>
<td>Compression strength, ( F_c )</td>
</tr>
<tr>
<td>Bending strength, ( F_y )</td>
</tr>
<tr>
<td>Bending strength, ( F_z )</td>
</tr>
<tr>
<td>Tension strength, ( F_t )</td>
</tr>
<tr>
<td>Stability coef., ( c )</td>
</tr>
</tbody>
</table>

The output for this case is shown below:

**BEAM STRESSES - Axis end (SELECTED)**

<table>
<thead>
<tr>
<th>Element</th>
<th>Node</th>
<th>( F_x/A )</th>
<th>( F_y/A^* )</th>
<th>( F_z/A^* )</th>
<th>( M_x/St )</th>
<th>( M_y/Sy )</th>
<th>( M_z/Sz )</th>
</tr>
</thead>
<tbody>
<tr>
<td>S5<em>6</em>12</td>
<td>6</td>
<td>-171</td>
<td>-11</td>
<td>17</td>
<td>0</td>
<td>1234</td>
<td>353</td>
</tr>
</tbody>
</table>

In summary:

\[
\begin{align*}
R_0 &= 0.223 \text{ (para 3.7.1)} \\
R_1 &= \text{Max}(0.15, 0.467) = 0.467 \text{ (para 3.3.3)} \\
R_2 &= 0.796 \text{ (para 3.9.2)} \\
\end{align*}
\]
R3 = 0  
Rmax = 0.796 (para 3.9.2 compression)

**SOLUTION CASE 2: REMOVE THE WIND LOAD**

Load only with Dead and Snow portions of load and set the duration factor Cd to 1.15.

\[
\begin{align*}
R0 &= 0.230 \quad \text{(para 3.7.1)} \\
R1 &= 0.652 \quad \text{(para 3.3.3)} \\
R2 &= 0.876 \quad \text{(para 3.9.2)} \\
R3 &= 0 \\
Rmax &= 0.876 \quad \text{(para 3.9.2)}
\end{align*}
\]

**SOLUTION CASE 3: DEAD WEIGHT ONLY**

Load only with Dead portions of load and set the duration factor Cd to 0.9.

\[
\begin{align*}
R0 &= 0.081 \quad \text{(para 3.7.1)} \\
R1 &= 0.278 \quad \text{(para 3.3.3)} \\
R2 &= 0.305 \quad \text{(para 3.9.2)} \\
R3 &= 0 \\
Rmax &= 0.305 \quad \text{(para 3.9.2)}
\end{align*}
\]

The wood geodesic dome model *Dome3V.fem* described later is a more practical use of the wood code checking procedure for a full construction project.

**COMPOSITEBEAM.FEM (COMPOSITE BEAM SECTION)**

This is an example of a composite beam. It is adapted from the CADRE Profiler (Section property design application) User Manual. CADRE Profiler can produce 'equivalent' section properties for multiple material cross sections.

In the CADRE Profiler application the main purpose of the exercise was to understand how to set up and determine equivalent section properties for composite beams. Here, the same example is used with those equivalent section properties to develop actual displacements, loads, and stresses at the material boundaries.

The specific task is to determine the stress in each material, find the ratio of stress to strength of each material and finally to conclude which material limits the beam's moment bearing capacity.

**MATERIAL ALLOWABLES**

The maximum allowable stress for the materials is given as:

- Steel 18000 psi
- Aluminum = 12000 psi
- Wood 1500 psi

**DESCRIPTION**

The beam is 100 inches long and simply supported. It is easily constructed with the quick modeler template with 20 segments.
Set up the simply supported 100 inch long beam with 10000 pounds down at the center, this will give a maximum bending moment of 250000 inch pound at the center of the beam.

The beam section is 3 inches wide. The top layer is 1 inch deep of steel. The middle layer is 6 inches deep of wood. The bottom layer is 2 inches deep of aluminum.

**COMPOSITE SECTION PROPERTIES**

The elastic modulus ratios for aluminum and steel relative to wood are given as 6.667 for aluminum and 20 for steel.

- Wood = 1 (i.e. E wood = 1.5E6)
- Aluminum = 6.667 (i.e. E alum = 10E6)
- Steel = 20 (i.e. E steel = 30E6)

Solving for the section properties by classical methods (or using CADRE Profiler) relative to wood as the reference material gives the equivalent section properties as:

- Area = 118 sq inches
- Iz = 1426.15
- Iy = 88.500

Extreme edge distances of the full beam to the top and bottom from the beam centroid are:

- CY1 = 3.5763
- CY2 = -5.4237

The effective area is 118 sq inches, while the actual physical area is 27 sq inches. This is the equivalent area of the section as if it were all wood. There are 18 sq inches of wood, 6 sq in of aluminum and 3 sq in of steel but the aluminum and steel are weighted.

\[ A = 18 + 6 \times 6.667 + 3 \times 20 = 118 \text{ sq in} \]

**USING EQUIVALENT STIFFNESS PROPERTIES**

For this example the stiffness properties that are needed are E (for wood 1.5E6), A, Iy, Iz, J. The parameters J and Iy can be arbitrary because we aren’t looking at torsion and side bending in this example. When you use equivalent stiffness properties in a finite element program, you always use them along with the elastic properties of the base reference material. So, the actual model and its stiffness are set up with the wood equivalent stiffness properties including the elastic modulus for wood (since that was our chosen reference). The final internal loads and displacements (and stresses) would be the same no matter if we had chose to model it with reference to aluminum, steel, or wood.

The effective stiffness properties for the beam, based on wood as the base would look like this.

**USING EQUIVALENT STRESS PROPERTIES**

Since side bending and torsion are not of interest, set SY to zero, ST to zero, and use 1.0 for both AZF and AYF. You will calculate Sz values at points of interest and by adjusting for the material at those points.

\[ S_z = \left( I_z / C_y \right) / R \]

Section stress moduli Sz must be determined at locations of interest.

The locations of interest are thefarthest edges of each material from the beam centroid.

- The steel extreme fiber distance (on top of the beam) is 3.5763 inches above the centroid. The elastic ratio is 20 from above.
- The aluminum extreme fiber distance (on bottom of the beam) is 5.4237 inches below the centroid. The elastic ratio is 6.67 from above.
- The wood extreme fiber distance (centroid down to the lower wood/aluminum boundary) is 3.4237 inches below the centroid (i.e. 5.4237 - 2.0). The elastic ratio is 1.0 since wood is the base material.
The section stress moduli for each location are:

- $S_Z$ for steel at the top of the beam = \[(1426.15/3.5763)/(20) = 19.9389\]. Compare the stress read here to the actual steel strength units.
- $S_Z$ for aluminum at the bottom of the section = \[(1426.15/5.4237)/(6.667) = 39.4402\]. Compare the stress read here to actual aluminum strength allowable.
- $S_Z$ wood at the wood/aluminum boundary = \[(1426.15/3.4237)/(1.0) = 416.55\]. Compare the stress read here to actual wood strength allowable.

If we set $S_Z$ in the Element Library with the steel value above (19.9389), we would read stresses as if it were all modeled with steel and we would check the result against the steel strength allowable. This would be the stress in the steel on the very top extreme fiber of the beam.

Similarly if we set $S_Z$ in the Library with the aluminum value (39.4402) or the wood value (416.55) above, the stress results would be for the aluminum extreme fiber location on the bottom of the beam, or the wood extreme fiber location at the lower wood/aluminum boundary respectively.

**STRESS ANALYSIS AT POINTS OF INTEREST**

For convenience you can set up 3 copies of the beam section in the Library with the only difference being the $S_Z$ value employed in each one. Then just select and assign them one at a time and check stresses (normal stress $S_{n1}$) at the axis node of the beam segment just to the left of center.

<table>
<thead>
<tr>
<th>Material</th>
<th>$S_Z$</th>
<th>$S_{n1}$</th>
<th>Allow</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>19.9389</td>
<td>12538</td>
<td>18000</td>
<td>0.696</td>
</tr>
<tr>
<td>Aluminum</td>
<td>39.4402</td>
<td>6339</td>
<td>12000</td>
<td>0.528</td>
</tr>
<tr>
<td>Wood</td>
<td>416.55</td>
<td>600.2</td>
<td>1500</td>
<td>0.400</td>
</tr>
</tbody>
</table>

Steel is clearly the limiting material in the beam because of the higher stress ratio.

The main point is that, for composite beams, the stress moduli ($S_Y$ and/or $S_Z$) must be set up taking into account the precise location of the extreme fiber point of interest in the section as well as the material elastic ratio with respect to the reference material.
COMTOWER.FEM (Wind loads on open frame structures)

The detail construction of this tower is the subject of a CADRE Analytic white paper #001C posted on the CADRE Analytic web site. The white papers are available at https://www.cadreanalytic.com for evaluators as well as licensed users.

The paper guides the user through the details of constructing this type of model using CADRE Pro. The construction methods involve using basic templates, replicating, tapering, and other construction tools to arrive at the final model.

Even if you aren't interested in towers, the construction techniques employed in the process are adaptable to many other types of models. The model is constructed so that it can be adjusted for alternate means of connections at the X bracing crossover. As presented, the crossover connection is made to be continuous on one member and pinned for the opposite leg of the X.

It is a simple matter to revise the X bracing pinned settings in the Element Library for any arrangement of pin fixity from 0 to 1 for either X direction based on the expected rigidity of the actual attachment hardware.

The white paper #001 also guides the user through methods of applying wind loads to this type of open frame structure without walls where the wind passes through the structure. The wind load discussion is summarized below.

Wind loads can be conducted on towers and other open lattice structures easily with CADRE Pro.

For these types of open lattice structures, it is useful to set each value of $F/L$ in the element library to the diameter (or width) dimension of the member so that the load, as initially determined, would be a unit loading (1 psi in this case). Then you can factor the load set or load sets to a value consistent with the wind speed and pressure coefficients.

This also provides a fast way of calculating the solid area of the tower since a projected unit load (1.0 psi) would give the projected area on the global plane (or any chosen plane) for that tube.

If we had a mixture of beam shapes, or needed to account for icing, then each member would be given an ‘effective’ diameter or width.

For the members called tubes, the outside diameter is the same as the specified diameter. However, for standard schedule 40 pipes, the outside diameter is larger than the specification:

<table>
<thead>
<tr>
<th>Pipe size</th>
<th>OD</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>8.63</td>
</tr>
<tr>
<td>6</td>
<td>6.63</td>
</tr>
<tr>
<td>4</td>
<td>4.50</td>
</tr>
</tbody>
</table>
In the element library, set each value of \( F/L \) to the outside diameter of the tube or pipe as indicated.

**WIND LOAD SPECIFICATION**

For illustration, the tower will be evaluated with a 100 mph wind condition under exposure condition C.

The wind velocity pressure is calculated taking into account exposure, terrain, and elevation

\[
Q \text{ (in psf)} = 0.00256\times K_z \times K_{zt} \times K_d \times V^2
\]

Where:

- \( K_z \) is an elevation factor dependent on exposure
- \( K_{zt} \) is a topographic factor
- \( K_d \) is a directional factor

For the preliminary simple checks assume that \( K_z \) is taken at the mean height of the tower (125 ft) and that all the other factors (\( K_{zt}, K_d \)) are 1.0.

\( K_z = 1.32 \) at 125 feet (Ref. ASCE 7-10, Table 27.3-1 at Exposure C)

\[
Q \text{ (in psf)} = 0.00256 \times (1.32) \times V^2
\]

\[ Q = 33.79 \text{ psf or 0.234 psi} \]

**SIMPLIFIED WIND LOAD METHOD FROM ASCE 7-10**

First, calculate the tower wind load using the simplified criteria in ASCE 7-10 for comparison purposes and to build confidence in the CADRE Pro methods. It is always important to have a general idea of the expected load levels before relying on the results of a complex finite element model.

ASCE 7 has specific simplified criteria for triangular towers. ASCE 7-10 (Fig. 29.5-3) gives the following gross pressure coefficient, \( C_f \), of a 3 legged tower constructed with round members:

\[
C_f = (3.4\epsilon^2 - 4.7\epsilon + 3.4) \times (0.51 \epsilon^2 + 0.57)
\]

Where \( \epsilon \) is the solidity of a single face of the tower (ratio of solid area to total area)

You can use CADRE Pro as a measuring tool to derive some of the basic geometric data needed for this classical analysis.

The simplified method is based on a single face of the tower.

If you have the white paper and the associated models that are provided with it, there is a single face model available for use (towerside.fem).
To make a face-only model, rotate the model from the reset position to Y=30; Z=1, then you can select the entire left face "on edge" with the frame.

With the side selected, use File/Save selected and save it as Towerside.fem.

Load the model Towerside.fem. From reset, rotate the model on Y to 120 degrees. Use Utilities/Constructs/Reorient which will set this position as the new global reference with Z essentially normal to the face. This is assuming we want the wind loads from a wind in the positive Z direction (out of the page). Now you have a single face to work with.

Measure the face solid area

Note: Ensure that the F/L values for all members in the element library have been set to reflect the outside diameter of the tubular member. (If they weren't tubes you would use the 'effective width of the member relative to the wind direction'.

Without selecting any element, right-click and use 'Set loads'. Open the Projected tab. Choose the Z direction, Positive, Replace, and then use 'Apply projected load'.

The value for Fz shown on the dialog is 68593 and, since we set F/L as the effective projected width (diameter) of the members, this value is the same as the solid area, Af, of the members in the single face as projected on the XY plane, normal to the wind.

Measure the face gross area

Next select the four nodes surrounding the face (2 at the bottom and 2 at the top) in a counterclockwise fashion. Then use the measure tool and read the enclosed area.

This is approximately the gross area of the face, Ag = 550393 sq in.

Since the sides aren't exactly straight, you could do this measurement more precisely using several tier measurements and adding them together but this is adequate for this particular tower for our purposes.

Calculate the wind force

The solidity, \( \epsilon \), of a single side is the ratio of Af to Ag or

\[
\epsilon = \frac{550393}{68594} = 0.1246
\]

ASCE 7-10 (Fig. 29.5-3) gives the following for Cf with round members

\[
Cf = (3.4\epsilon^2 - 4.7\epsilon + 3.4) x (0.51 \epsilon^2 + 0.57) = (2.8671) x (0.5779) = 1.657
\]

Wind force = Cf x Q x Af = (1.657)(0.235)(68593)
Wind force = 26709 pounds.

Although we used CADRE Pro as an aid to calculate some of the geometric values, the above result is just the typical simplified analysis performed according to the simplified tower criteria in ASCE 7-10.

This gives the analyst an estimated gross value of lateral force due to the wind load using a 1500 foot level for wind pressure.

CADRE PRO ANALYSIS FOR COMPARISON

Next, a wind analysis will be made with CADRE Pro in a manner that allows us to compare directly with the simplified criteria above using the same elevation assumption for determining wind pressure.

You can do the complete analysis, including distribution in CADRE Pro with wind from any direction. This would be an analysis using the full model, not just a face.

Use the full sample model ComTower.fem:

Technical data, including that in ASCE 7-10 for open lattice signs (Fig. 29.5-2), indicates that a reasonable pressure coefficient for individual round tubes under these wind conditions would be

\[ C_p = 0.9 \]

As determined above, the 100 mph wind pressure is 0.235 psi in exposure C at 1500 feet elevation.

\[ C_p \times Q = 0.234 \times 0.9 = 0.2115 \]

Without selecting any elements or nodes, use Set loads. Open the Projected tab, Z direction, Positive, Replace and use Apply projected load.

The value of Fz shown is

\[ F_z = 127868 \]

This is force under a 1.0 psi level of load or, more exactly, just the total area of each member of the full tower (all three sides) projected on the XY plane. The actual wind load would be this value multiplied by \( C_{pq} \) above.

\[ F_z = 127868 \times 0.2115 = 27044 \text{ pounds.} \]

So, before leaving the dialog, open the File tab on the dialog. Press the 'Factor' button and enter a value of 0.2115 in the field and accept.

The gross applied load on the model is now

\[ F_z = 27044 \]

Press OK to leave the global loading dialog.

The 27044 pound wind load is rationally distributed over the tower.

Compare the full tower analysis 27044 with 26709 from the simplified method.

FULL ANALYSIS WITH VARIABLE WIND SPEED VS ELEVATION

The analyses illustrated above were mainly to build confidence and provide the analyst with a reasonable level of loads to expect from the more comprehensive analyses typically needed for an open-framed tower.

The analysis of a tower is usually conducted with the wind pressure adjusted for each elevation rather than as a single average and it is also derived for multiple directions.

Assign group names

To this end, one can select and provide group names for various levels up the tower. About 10 to 12 levels would be normal for this tower, but, for simplicity of this presentation, we specified our wind elevation segments at the same 7 elevations that were used for element library property definitions.
You can use ASCE 7-10, Table 27.3-1 at Exposure C to develop a table and derive the value of wind pressure at each tier (taken at the mid level of each tier).

<table>
<thead>
<tr>
<th>Tier</th>
<th>Elev.</th>
<th>Ave(in)</th>
<th>Ave(ft)</th>
<th>KZ</th>
<th>QKZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0-480</td>
<td>240</td>
<td>20</td>
<td>0.9</td>
<td>0.144</td>
</tr>
<tr>
<td>2</td>
<td>480-1200</td>
<td>840</td>
<td>70</td>
<td>1.17</td>
<td>0.1872</td>
</tr>
<tr>
<td>3</td>
<td>1200-1680</td>
<td>1440</td>
<td>120</td>
<td>1.32</td>
<td>0.2112</td>
</tr>
<tr>
<td>4</td>
<td>1680-2160</td>
<td>1920</td>
<td>160</td>
<td>1.4</td>
<td>0.224</td>
</tr>
<tr>
<td>5</td>
<td>2160-2400</td>
<td>2280</td>
<td>190</td>
<td>1.45</td>
<td>0.232</td>
</tr>
<tr>
<td>6</td>
<td>2400-2640</td>
<td>2520</td>
<td>210</td>
<td>1.48</td>
<td>0.2368</td>
</tr>
<tr>
<td>7</td>
<td>2640-3007</td>
<td>2823.5</td>
<td>235.29</td>
<td>1.52</td>
<td>0.2432</td>
</tr>
</tbody>
</table>

Redefine group names of elements according to the elevation. They have been assigned as TIER1 to TIER7 on the sample model.

The F/L property in the library should contain the outside diameter (or effective width) of each beam member.

Develop unit load sets:

Organize folders in your project with a Loads folder and two unit loads subfolders as follows:

loads
  UnitX
  UnitZ

Next, develop a set of unit loads for wind in the X direction and another unit load set in the Z direction. Then, use the assembler to form the two unit load sets into wind load cases. These two directions chosen include one normal to a face and one parallel to a face which are probably all that would be needed for a triangular tower unless there were surrounding terrain issues to address.

Outside of any editing mode and with nothing selected, right-click and use Set loads.

Open the Projected tab. Choose the X direction, Positive. Choose Beams only.

Click on the Unit sets button.

In the dialog, choose all 7 tiers, and check the item to Use beam F/L property as the beam width.

This assumes you have set the outside diameter as the F/L property for all beam definitions in the element library.

In the next query dialog, browse to choose the UnitX folder as the place to collect the unit load sets for the X direction.

Press OK and you should get confirmation that 7 files were created and stored as requested.

That completes the unit sets for the X direction.

Choose the Z direction on the Projected tab and repeat the process above using the UnitZ folder to store those unit set loads.

Assemble the final load sets

While still in the element library, open the File tab and click on the Assemble button. Click on the Folder button on the assembler dialog and navigate to the UnitX folder.

Change the header name “Case 01” to “WindX” and enter the pressure value QKZ for each tier as shown in the above table.

Next save the load set and also save the dialog configuration setup in case you need to return.

Use Save configuration to save the setup. Save the configuration file inside the UnitX folder. Name it WindX.cfg.

Use Save cases to save the load set. Save the load case set outside the unitX folder under the Loads folder. The load set is named after the column header, WindX.lsb.
You can immediately create the load set for the Z direction. Just click on the Folder button, and navigate to the UnitZ folder. Rename the header to “WindZ” and fill in the pressures which are the same as before.

Use Save configuration to save the setup. Save the configuration file inside the UnitZ folder. Name it WindZ.cfg.

Use Save cases to save the load set. Save the load case set outside the unitZ folder under the Loads folder. The load set is named after the column header, WindZ.lsb.

Now you have complete full tower load sets saved for both the X and Z directions.

Exit with OK which returns you back to the File tab on the Global loading dialog.

The current load set from the selected column WindZ will be loaded onto the model.

The gross tower load in the Z direction is 24620 pounds.

Use the Get button and load the WindX case.

The gross tower load in the Z direction is 24338 pounds.

Notice that the loads are a little lower than the preliminary analyses which used a single elevation at the mean height. This is to be expected since the more rational analysis takes into account that the larger members are at the lower elevation where pressures are lower and the smaller members are at higher elevation in the greater wind speed.

The sample model is delivered with Wind X merged with the tower dead weight.

**CONTACTBEAM.FEM (BEAM ON CONTACT SUPPORTS)**

CONTINUOUS BEAM ON CONTACT SUPPORTS

This model illustrates the construction of contact supports using one way nodes which are active in one specified direction but released in the other.

This is a 1000 inch long AISC M 6 x 4.4 beam resting (but not attached) on five supports. For this example the beam is considered weightless, although adding the dead weight would be a simple matter. The beam is allowed to lift off the supports wherever it would naturally tend to do so.

The section properties for the beam can be extracted directly from the Steel USA database and imported to the element library.

The supports are one-way fixed nodes that have been set up to be active with a positive reaction force but release whenever the reaction would be negative. The one-way designation is set up in the Boundary Assignment Dialog by choosing "Fixed", checking the one-way box, and then choosing the direction of reaction force (positive or negative)
for which the fixed reaction is "active". In this case, with Y positive (+), the reaction point is positive when the reaction vector points in the positive Y direction.

The dashed horizontal 'Line element' is drawn across the support locations attached only on the ends to other independent nodes. It is only for visual purposes in this illustration.

To check the lift off after solution, select the end node at the right end of the beam. The value is 9.215 inches upward at that end. The only reaction vectors active are those pointed upward in the positive Y direction. Other reactions are inactive and suppressed in the display of results although they still exist as part of the unsolved model definition.

Many of the intermediate nodes have been hidden to improve visibility of the important aspects. To see all features, turn on the show/hide tool on the tool bar.

Classical solution for comparison:

\[ Y = \frac{L_3 (Pa)}{(6EI(L_2^2 - a^2))} \]

Where:

\[ P = \text{concentrated load} = 1000 \]
\[ a = \text{distance node 2 to load } P = 120 \]
\[ EI = E \times I = 20.88E7 \]
\[ L_1 = \text{distance node 1 to node 2} = 250 \]
\[ L_2 = \text{distance node 2 to node 3} = 250 \]
\[ L_3 = \text{distance node 3 to node 5} = 500 \]
\[ Y = 9.214559 \]

To examine the behavior of the lift off nodes move the load from its current position and place it approximately at the midpoint between the two reaction points on the right and re-solve.
CONTACTNODES.FEM (BEAM ON CONTACT SUPPORTS- STABILITY ISSUES)

This is a 1000 unit long beam on 10 one-way ('contact') supports.

The beam is an AISC M 6 x 4.4 steel.

The two end supports are set with the Y global degree fixed against upward movement but free to move down. The space between reaction points and loads are all 100 inches.

The other 8 reactions are set with Y fixed against downward movement but free to move upward.

The center node is not intended to be restrained.

The center node is loaded with 1000 pounds downward.

Solve and you will get the following correct solution.

STABILITY ANOMALIES

Now reverse the load in the middle so that it is 1000 pounds upward.

This should solve with by lifting upward off the 8 supports while being restrained at the ends.

However, it doesn't solve correctly because of convergence and instability issues along the way to a solution.

RESOLUTION

Select the center node (or any other non-bound node) and set a boundary condition there as a SPRING type in all 6 degrees of freedom with a VERY SOFT spring (so soft as not to have any effect on numerical accuracy).

Try K = 0.001 pounds/inch which is ver soft considering the 1000 pound load and the size of the beam.

X spring K = 0.00001
Y spring K = 0.00001
Z spring K = 0.00001
Xrot spring K = 0.00001
Yrot spring K = 0.00001
Zrot spring K = 0.00001

(Only Y and Zrot are really essential for this particular model)
Now is solves correctly.

This stability anomaly rarely occurs on real structures but is rather common on simple beam models with only a few discrete nodes. It can easily be resolved by strategic stabilization with a soft spring without affecting results.

Typically, if the structure would be stable with all the one-way boundary conditions set to "free" then one doesn't need to place any dummy soft stabilizing boundary on the model.

**CONTACTPLATE.FEM (PLATE ON CONTACT SURFACE)**

This illustrates the method of configuring contact nodes on 3D plates to provide support yet allow lift off as required. This is the type of modeling that might be necessary for base plates or brackets mounted on elastic foundations.

The model is a simple "L" bracket made of AISI 301 stainless steel. Units are millimeter and Newton (Mpa system)

The base flange, resting on a rigid substructure, is 96 mm x 96 mm and the vertical flange is 48 mm high. It is actually 3D but viewed in these pictures from the edge.

The bracket is pinned at both the left and right edges and loaded at the upper edge of the vertical flange is loaded with an outward load of 10 Newton distributed along the edge.

The left edge of the base is pinned (X, Y, and Z fixed) to the substructure allowing rotation. The right edge is pinned but allowed to slip in the X direction. (i.e. X free, Y fixed, Z fixed, Xrot fixed, Yrot fixed, Zrot fixed).

All the remaining nodes in the base are free except that the Y direction is fixed with a one-way specification allowing lift off if it would tend to do so. This is set by specifying that the Y fixed restraint is active only when the Y reaction force would be positive.
Solving with the load oriented to the right results in a lifting of the base from the substructure.

Reverse the load. To reverse the load do not select anything, right-click on the screen and choose *Set loads*. On the *File* tab, choose the *Factor* button and place -1 in the field provided. Use *OK* to reverse, and *OK* again to return to the model. The loads should now be pointing to the left.

The base plate is not allowed to displace beyond the rigid floor.

For an example of a beam on one-way supports see the sample file *contactbeam.fem*.

**COOLTOWER.FEM (COOLING TOWER)**

This is a cooling tower modeled under a wind load condition - See exercise 22 in Help "Getting Started" for step by step details of construction and solution.
BEZIER CURVE
This models starts with a simple Bezier curve in the shape of the vertical edge. The Bezier is made with two end nodes and two end-slope guide points.

REPLICATE (CLONE METHOD)
Replicate the line (radially WITH interconnectors) around the vertical axis (like a surface of revolution) to make a wire frame quadrilateral mesh.

CLAD
Clad the quadrilateral framework with plate elements, then select and delete the beam framework.

This sample illustrates the power of combining relative simple tools to make complex surface models.


CRANE1.FEM (SIMPLE CRANE STRUCTURE)
Advanced analysis and the use of the offset or eccentric beam elements (Exercise 14 in "Getting started" of Help)
This exercise in Help uses the standard style of eccentric beam where the end loading nodes are identical at each end of the member. An additional model Crane2.fem uses an alternate eccentric beam style with different moments on each end of the beam element.

PROPERTIES
BOOM - Steel T - section (eccentric beam element)
width = 6.0
death = 4.0
web & flanges = 0.375
radii = 0.375
E modulus = 29000000
G modulus = 11420000
Density = 0.283
Area = 3.66973E+00
W/L = 1.038534 lb/in

SUPPORT CABLE - wire rope (tension beam element)
6 x 19 standard wire rope
D = 5/8 in diameter
Nominal area = 0.3068
Metal area = 0.38 x D^2 = 0.1484 sq in
Elastic modulus = 1.2E7 psi on nominal area  
W/L = 0.052 lb/in

With this example both the load point and the wall attachment point are 6 inches above the boom neutral axis. When the loading (and reaction) eccentricity is the same for both ends of an element or member you can use the 'Standard' style of eccentric beam.

An alternate model Crane2.fem demonstrates alternative styles of the eccentric beam with different end load offsets. To build the model from scratch, first set 3 points.

1) 0, 0, 0 (load point)
2) 240, 0, 0 (boom-wall attachment)
3) 240, 138.564, 0 (cable attachment)

Draw beam type elements from 1 to 2 (to become the boom) and from 1 to 3 (to become the cable).

ELEMENT LIBRARY ENTRIES

In the element library you can set up the boom "T" property and the cable wire property as described above.

THE HORIZONTAL BOOM

You T section dimensions are shown below.
You can conveniently use the 'Basic Shapes' feature to set up the T properties.

After importing, set the boom property type to *Eccentric beam* then set its 'style' to *Standard*.

Set the offset magnitudes to "Dy = 6" and "Dz = 0".

Since Dz is zero, this indicates the load points on the eccentric beam at both the origin and axis nodes in are 6 inches directly above the neutral axis of the beam (which we are assuming to be horizontal).

The wire rope

Set up the steel wire 6 x 19 wire rope in the library.

There are two ways of handling wire rope,

1. Use the full elastic modulus of steel along with the area of steel in the rope.
2. Use a reduced elastic modulus along with the nominal area of the wire rope.

Specifications vary as to the precise values to use which are often established by tests. In any case, you must be consistent in using a cross section area directly associated with the published elastic modulus. Here we assume that we have a specification that provides an elastic modulus and density based on nominal area.

When using input properties based on nominal area, be sure to interpret the output stress in the wire rope also relative to nominal area. In that regard, the allowable strengths are per unit nominal area, or convert them to stress per metal area. The area of metal is provided in most specifications.
There is a database of wire rope in the section property database for American steel which you can use to import the properties directly for the 5/8 6 x 19 hoisting rope.

Use the Database tool and import the property Wire rope collection in the American steel section database or just use the Add button and enter the properties directly.

Either way the values should be:

- \( E = \) elastic modulus = 1.2E7 psi (relative to nominal area)
- \( G = \) shear modulus = 4600000 psi (relative to nominal area)
- \( \frac{W}{L} = (0.1698)(0.3068) = 0.052 \text{ lb/in} \)
- Area = 0.3068 (nominal)

If you directly added the values then type a name such as \textit{Wire rope 6 x 199 Standard 5/8 inch}

In the element type drop down list select \textit{Tension beam}.

Set \( I_y, I_z \) and \( J \) to zero.

Note: For a tension type element, if zero values are set for the \( I_y \) and \( I_z \) then CADRE Pro internally uses a value based on a diameter that is 1/50000th of the length.

That should be small enough in our model not to induce any significant bending forces at the ends.

You have two entries in the library at this point. Exit the library.

**ASSIGN PROPERTIES**

Open the Element Editor mode.

Select the cable element and use the 'properties' button to assign the cable properties to the cable element on the model.
Select the boom and assign the "T" eccentric beam property to the boom element. The boom will now take on the shape of the eccentric beam with the outer end load point 6 inches above the neutral axis and the neutral axis should be horizontal.

DISCRETIZE

The 'divide' operation in CADRE Pro is a smart divider. It will divide an element with an appropriate transition from one end to the other provided the properties are already assigned.

In the Element Editor mode, select the boom. Right-click and from the pop-up menu choose "Divide". Divide the boom into 8 equal parts.

LOAD

Nodal editor mode. Select node 1 and use 'Set loads' to set a 5000 pound load for Fy, all other degrees of loading to be zero.

RESTRAINTS

You can fully fix node 3 since it is a wire cable and the fully fixed condition won't matter much. For the boom connection to the wall, select node 2 and set all degrees of freedom to fixed except Zrot which should be left free.

To investigate 2D bending and stability in the XY plane you can lock out potential lateral stability issues by setting some restraints on node 1. Fix Z, and Fix Xrot while all else remain free is a good way to do this.

SOLVE

Solve with Standard solution and Static advanced solution (with 1 increment only!)

Notice the difference resulting from the p-delta effect.

Check node 7 displacement at the beam center with standard solution: Y = -3.8468
Check node 7 displacement with advanced solution with one increment: Y = -5.5116

All of the additional deformation is due to end moments since no dead weight or distributed loads are applied over the boom span. One could easily add the dead weight effect using the global dead weight distribution function.

CRANE2.FEM (SIMPLE CRANE STRUCTURE)

Advanced analysis and the use of the offset or eccentric beam elements (Exercise 14 in "Getting started" of Help)

This second example uses the Offset Y style of eccentric beam where the end loading nodes are unequal at each end of the member. See: Exercise 14 of "Getting started".

PROPERTIES

BOOM - Steel T - section

width = 6.0
depth = 4.0
web & flanges = 0.375
radii = 0.375
E modulus = 29000000
G modulus = 11420000
Density = 0.283
Area = 3.66973E+00
WL = 1.038534 lb/in

SUPPORT CABLE - wire rope
The wire rope can property can be imported directly from the Steel USA database. Choose the 6 x 19 x 5/8 hoisting rope type. Or just enter the properties directly.

Standard wire rope, 6 x 19, 5/8 in diameter
Nominal area = 0.3068
Metal area = 0.38 x D^2 = 0.1484 sq in
Elastic modulus = 1.2E7 psi on nominal area
W/L = 0.052 lb/in

The transition from the 6 inch offset at one end to no offset at the other is accommodated by employing one of the alternate eccentric beam styles; in this case Y Offset. This alternate style always lies within the local XY plane of the element's local coordinate system with the offset of the load from the neutral axis in the +Y direction. However, with this style, the offset magnitude can differ between the two ends.

To build the model from scratch, start a new model with width of 240 and height of 140.

Then set 3 points.

1) 0, 0, 0 (load point at the outer end of the boom)
2) 240, -6, 0 (boom-wall attachment)
3) 240, 138.564, 0 (cable attachment)

Draw beam type elements from 1 to 2 (to become the boom) and from 1 to 3 (to become the cable).
ELEMENT LIBRARY ENTRIES

In the element library you can set up the boom "T" property and the cable wire properties as described above. You can conveniently use the 'Basic Shapes' feature to set up the T properties.

Set the boom property type to *Eccentric beam* then set its 'style' to 'Y Offset'.

Set the offset magnitudes to "Dyo = 6" and "Dya = 0".

This indicates the load point on the eccentric beam at the origin in is 6 inches above the neutral axis of the beam (which we are assuming to be horizontal and even with node 2 at the wall).
Add the wire rope property by directly entering the data, or by importing from the other model Crane1.fem, or directly from the USA Steel database as 5/8 diameter 6 x 19 hoisting rope.

Set the wire rope type to *Tension beam*.

You have two entries in the library at this point. Exit the library.

**ASSIGN PROPERTIES**

Open the Element Editor mode.

Select the cable element and use the 'properties' button to assign the cable properties to the cable element on the model.

Select the boom and assign the "T" eccentric beam property to the boom element. The boom will now take on the shape of the eccentric beam with the outer end load point 6 inches above the neutral axis and the neutral axis should be horizontal.

**DISCRETIZE**

The 'divide' operation in CADRE Pro is a smart divider. It will divide an element with an appropriate transition from one end to the other provided the properties are previously assigned.
In the Element Editor mode, select the boom. Right-click and from the pop-up menu choose "Divide". Divide the boom into 8 equal parts.

The boom horizontal line follows the neutral axis of the boom while the nodes generally locate the load path along the boom with gradually diminishing offsets.

If you check the element library at this point you will notice several more entries corresponding to each of the new configured segments on the boom.

LOAD

Go to the Nodal editor mode. Select node 1 and use 'Set loads' to set a -5000 pound load for Fy, all other degrees of loading to be zero.

RESTRAINTS

You can fully fix node 3. Since it is a wire cable, the fully fixed condition won't matter much. For the boom connection to the wall, select node 2 and set all degrees of freedom to fixed except Zrot which should be left free.

To investigate 2D bending and stability in the XY plane you can lock out potential lateral stability issues by setting some restraints on node 1. Fix Z, and Fix Xrot while all else remain free is a good way to do this.

SOLVE

Solve both with Standard solution and with 1 increment of Advanced solution and see the difference in displacement at the middle part of the beam (p-delta effect).

Check node 6 displacement at the beam center with standard solution: Y = -2.335
Check node 6 displacement with advanced solution with one increment: Y = -3.009

You can check select all nodes and use Results/Displacement diagram to generate a plot of displacement across the beam and confirm that node 6 is the largest displacement. If you previously save the two results (advanced and standard) you can plot both with the envelope on the same plot.

One can build a simple model just using standard beams by employing a relatively rigid short stub beam on the end to make the offset. With a short stub 100 times more rigid than the boom such a model would provide a condition
number of 9.7 while this eccentric beam model has a condition number of 13.7 (far more accurate). The stub approach would be acceptable here but as the stub gets shorter and the model becomes more complex, the use of an eccentric beam in place of short stubs may become essential.

**CURVEDBEAM.FEM (Curved cantilever beam)**

**LOADED NORMAL TO THE PLANE OF CURVATURE**

This is a textbook example used to illustrate curved beams with a load normal to the plane of curvature. The diameter of the solid stainless steel shaft is 1 inch and it is loaded with 100 lb normal to the plane. The results are compared to the theoretical solution derived from Roark *Formulas for stress and strain*.

**CONSTRUCTION**

Set points A, B, C

A = (24*Cos (150), 24*Sin (150), 0) = (-20.78, 12, 0)
B = (24, 0, 0)
C = (0, 0, 0)

Select points C, A, B in that order

Use Utilities/Insert special structures

Choose Arcs and circles
Choose Arc < 180
Choose Sections = 32

Restrain the embedded end A as fixed in all 6 degrees of freedom.

Set a load on the free end B as Fx = 0; Fy = 0; Fz = 100

**SOLUTION COMPARED THEORETICAL VALUES (ROARK)**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Z</td>
<td>5.191758</td>
<td>5.189681</td>
</tr>
<tr>
<td>Xrot</td>
<td>-0.101798</td>
<td>-0.101762</td>
</tr>
<tr>
<td>Yrot</td>
<td>-0.099093</td>
<td>-0.099086</td>
</tr>
<tr>
<td>Reactions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rz</td>
<td>-100.0</td>
<td>-100.0</td>
</tr>
<tr>
<td>Rxrot</td>
<td>1200.0</td>
<td>1200.0</td>
</tr>
<tr>
<td>Ryrot</td>
<td>-4478.5</td>
<td>-4478.5</td>
</tr>
</tbody>
</table>
CYLINDERSX.FEM (INTERSECTING CYLINDERS)

This model is two aluminum metric cylinders, one penetrating through the other. The vertical one is 250 mm in diameter, the horizontal one is 150 mm in diameter. Units: Millimeter/Newton system

This model is two 6061 aluminum metric size cylinders, one penetrating through the other. The vertical one is 250 mm in diameter, the horizontal one is 150 mm in diameter. Units: Millimeter/Newton system

This model was constructed entirely within CADRE Pro. This type of construction involving intersecting cylindrical surfaces is the subject of CADRE Pro white paper #12. The white papers are available at https://www.cadreanalytic.com.

The CADRE Pro features used are:

- Quick modeler
- Merge
- Clone
- Clad
- Cut hole
- Force to radius
- Force to angle
- Mirror

LOADING

Pressure inside small cylinder is 0.21 MPa

Pressure inside large cylinder is 0.07 Mpa

The small cylinder surface inside the larger is loaded with 0.14 internal (differential pressure)

Such intersecting curved surfaces can also be meshed with a CAD Modeler such as Rhino 3D but this shows that one can accomplish it in CADRE Pro.

You can use Model/Configuration and display the maximum and minimum aspect ratios for plate elements. Notice that the maximum value about 3 which is fine for reliable results in those areas.

An advantage of constructing it in CADRE Pro is that you can set up properties and group names as you go along so that when its finished, its finished, and you don't have to fish around in a large complex model to select and assign properties. Also, carefully engineered meshes constructed with CADRE Pro usually provide more reliable results than automatic meshing features and result in more efficient models that solve much faster with far less memory.

See white paper #12 "CONSTRUCTING MODELS WITH CYLINDRICAL INTERSECTIONS"
**CYLINDER45.FEM (NON-ORTHOGONAL CYLINDER INTERSECTION)**

The sample file CylindersX.fem concerns orthogonal intersecting cylinders that mate at 90 degrees. This sample involves a mating cylinder that attaches at an angle other than 90 degrees. In this case you first need establish a user defined system with its XY plane normal to the axis of the mating cylinder and the Z axis of the system parallel to the mating cylinder axis.

![45 Deg Sys]

The quickest way to create such a system is to draw a beam element from the common intersection node on the cylinder axis to the node where the mating cylinder axis penetrates the wall. Then, select the beam and right click and use **Define system** from the pop-up menu. With only one beam element selected there is a special feature that will automatically create a user defined system with its XY plane normal to the beam and its positive Z axis along the beam from the origin toward the axis node.

From this point on, all construction operations are the same as for the orthogonal connection except that on each dialog where the hole is configured you must select the user defined plane rather than one of the orthogonal Cartesian planes.

Special features used in this construction are:
- Constructs/Cut hole
- Constructs/Force to radius
- Constructs/Force to angle

See CADRE Analytic white paper #12 for details of constructing orthogonal and non-orthogonal intersecting cylinders. The white papers are available at [https://www.cadreanalytic.com](https://www.cadreanalytic.com).

**DEEPBEAM.FEM (DEEP BEAM)**

This sample is used to illustrate the use of the 'Deep beam' element where shear displacement is significant.

![Deep Beam]

The aluminum beam is 1 inch deep and 0.1 inch wide and 1 inch long. (Not a "slender beam") and loaded with 100 pounds at the end.

This model includes just the neutral axis of the beam.
To see an outline of the actual beam turn on the 'Extents' which have been set to outline the actual beam shape.

Another way to make the picture more representative is to increase the draw width. Change the draw width in the Element Library to 100. You can zoom in or out to shape the beam view.

These amenities are strictly visual and have no effect on results.

This example has 3 library entries that can be assigned and compared.

Standard (slender) beam for which the shear displacement is ignored

Deep beam with A/As = 1 (shear displacement considered but no reduction in shear area effectiveness is considered)

Deep beam with A/As = 1.2 (shear displacement using the published value for shear area effectiveness for a rectangular solid section)

THEORETICAL CALCULATIONS

$$Y = Y_{bending} + Y_{shear}$$

For a cantilever beam loaded at the end:

$$Y = Y_{bending} + Y_{shear} = \frac{PL^3}{3EI} + \frac{PL}{AsG}$$ where As is the effective shear area

$$Y_{bending} = \frac{PL^3}{3EI} = 4.0E-4$$

$$Y_{shear} (\text{case 1}) = 0$$

$$Y_{shear} (\text{case 2}) = 2.647E-4$$

$$Y_{shear} (\text{case 3}) = 3.175E-4$$

Total displacement Y:

Case 1: $Y = 4.0E-4$
Case 2: $Y = 6.647E-4$
Case 3: $Y = 7.175E-4$

CADRE Pro output is identical to these theoretical results.

COMMENTS AND CONCLUSIONS

1) The shear deformation is seldom necessary for "slender beams" but may become essential for non-slender ones.
2) Even when using a shear area equal to the actual area, there is still a significant contribution to displacement from shear if the beam is very short in comparison to its cross section.
3) The degree of discretization doesn't significantly affect this result even though the small segments are very short beams in comparison to the section. For a non-slender beam "member", you will get essentially the same correct answer with the deep beam no matter the degree of discretization and essentially the same wrong answer with a standard (slender) beam irrespective of the discretization.

3) There are published values of the ratio of $A/\text{As}$ for various sections.

4) The ratio $A/\text{As}$ (always $> 1.0$) is sometimes called the "Form factor" but this name is also used for several other factors related to cross section stress so there can be considerable confusion. That is why CADRE Pro requests the shear effective area directly rather than specifying a 'form factor' for input. See Help item on "Deep Beam Shape Factors".

5) Don't confuse the shear stress 'factors' $A_yf$ and $A_zf$ with the shear stiffness 'areas' $A_{sy}$ and $A_{sz}$. These latter ones are specifically shear stiffness parameters representing the area effective in resisting shear deformation. The other parameters are proportional factors on the gross area used for shear stress analysis of the section. They may require values less than 1.0 for some shapes depending on the stress quantity that is to be calculated.

**DISHANTENNA.FEM (Applying wind loads)**

This is a 120 inch diameter parabolic antenna with focus located at 50 inches from the center. The antenna is to have 12 spokes and 8 rings spaced evenly across the disk.

This sample is intended to demonstrate some approaches to wind loading as well as to illustrate use of certain templates and constructors including the surface shaping utility. A similar 'dynamic model' (AntennaSeismic.fem) of a different type of parabolic antenna demonstrates some aspects of seismic analysis.

![Antenna Diagram](image)

**GEOMETRY**

Focal length: $F = 50$

From analytical geometry the equation of surface is:

$$R^2 = 4FX$$

or:

$$X = 1/(4F)R^2 = 0.005R^2$$
DISH CONSTRUCTION

Use **Utilities/Quick modeler**

Choose:
- Spheric template
- ZY plane
- X,Y,Z = 0,0,0
- Radius 60; Height 0 (makes a flat mesh)
- Sections 8; Sectors 12; No. iso adjusted 0

Plates AND Beams

As created with the axis normal to the ZY plane you are looking at the surface from the edge. Press the **ZY** plane view tool button to look directly at the disk surface.

When created, the flat disk will already have group names assigned to components by the Quick modeler template: **Radial, Ring, Ring outer, Internal, and Plate**. Use **Select/By group** and select all the **Internal** elements. From the element editor mode, use the **Delete** item from the pop-up menu to delete all of the **Internal** beams leaving only the main radial arms and the main rings along with the plate surface.

Select all nodes
Use *Utilities/constructs/Shape surface*

Choose the equation for the polar parabola: $W = W + AR^2$

Set: $W = X$
Set: $M = 2$
Set: $A = 0.005$

Do NOT check the item First selected node is the reference point.

The default point (0, 0, 0) at the center will be used as the reference point.

This will make the precise parabolic shape with focus at 50 inches in the ZY plane (standing vertical and pointed horizontal to the right as shown below.)
SUPPORT STRUCTURE

The dish is supported from the two nodes at 7.5 inches horizontally on each side of the center node.

Select these attachment nodes then use ‘Copy nodes relative’. Set X = -9 and check the option to include connecting beams.

This creates the support arms extending 9 inches behind the disk from the support nodes. Give these two beams the group name ‘Antenna support’. The focal support is accomplished the same way using the center node and +50 inches. It is named ‘Focal support’.

ORIENTATION

The tip of the focal beam is used to orient all the radial spokes and the disk center node is used to orient all the radial beams in the disk. The support arms are reference to a ground or tower post node directly under each one. The focal arm is referenced to the lowest or highest node on the disk so that its larger cross section dimension is oriented vertically.

TOWER CONSTRUCTION

The tower can be made using simple standard construction methods. Briefly, the support arm free end nodes are selected and ‘Copy nodes relative’ is used (with connecting beams) to extend 2 posts 100 inches downward to the ground restraint point. Give them the group name; ‘Tower post’. The ‘Divide’ method is used to divide both posts into ten equal parts. Cross beams are drawn between the posts. The complete two post construction is selected and replicated 9 inches farther behind with connections at each segment creating the 4 post design. Finally, side cross diagonals are drawn in. You can now select all the ‘unnamed’ group elements and name them ‘Tower cross’ which includes all the horizontal and diagonal members.

ALTERNATE TOWER

If you are adept at using the merge utility (Utilities/Merge), you can make the tower separately in one operation using the ‘Box’ template.

First select one of the arm nodes on the top of the tower and use Copy nodes relative to copy it 10 units downward on the Y axis. This is to make a convenient reference node from the final merging step.

Save the dish part of the model.

Then open Utilities/Quick modeler, choose the Box template and set the following.

Corner node 1: 0, 0, 0
Corner node 2: 9, 100, 15
Once created, rename the group names for the posts and cross members as described above and then save the tower portion.

Reload the dish model and proceed to merge the tower onto the dish model.

On the dish model you would select the reference node you created and the two arm attach points in order (c, b, a). Then use **Utilities/Merge** and select the corresponding tower reference node and corresponding tower attach points in the same order (c, b, a).

![Diagram showing tower attach points](image)

This will attach the tower to the dish with the correct orientation. Accuracy might have been better if we had more distance to the reference node (c) such as at the full 100 units to the ground and at the tower ground. We used 10 units here just to keep the picture small.

**ANTENNA ELEVATION**

The antenna, as first constructed, is vertical and pointing horizontal to the right. To change elevation, all the elements (both plate and beam) in the antenna and support arms are selected along with a single node at the rotation point. It is easier to make this selection using Select/Elements/By group then select the one of the nodes at the end of the dish support arm on the tower. Then use 'Constructs/Rotate' and rotate about the Z axis. Here it is rotated by 30 degrees. You can easily re-perform this operation to reset the antenna for other elevations.

**ELEMENT PROPERTIES**

The tower is made of 2 diameters of steel tube while the antenna is made of 2 diameters of aluminum tube with a thin non-structural foil membrane. The support arms are a rectangular steel tube. The focal support beam is rectangular tube of polycarbonate material. These properties are set up in the element library and assigned to the tower members.

Each direction of wind would be considered separately. Here we illustrate a wind loading in the Z global direction across the dish curvature. (i.e. directed out of the screen in the initial startup position). Note: you can set up ‘defined coordinate systems’ to orient the loading in any direction desired.

**WIND IN THE Z DIRECTION**

Open frame structures and closed frame structures respond differently to wind forces. So, for these types of structures it is best to develop loads on the dish surface and on the open framework tower separately and then combine them to obtain the full wind load on the structure.

There are two general methods of loading, one is by selection of the elements and applying a user specified intensity, the other method is without selection and letting the F/L value is the library dictate the intensity. The later method works best on open frameworks. To illustrate both methods, we will use the later method to develop the framework loads (tower, support arms, and focal support, and then we use the former method (by selection) to develop the dish loads.

Assume that wind design pressure has been calculated using all appropriate factors to be \( p = 0.1 \) psi

**FRAMEWORK LOAD**

The wind force on a beam component is \( F = C_F \times p \times A \) where \( C_F \) is the pressure coefficient and \( A \) is the cross section area as projected normal to the wind.
The projected loading function in CADRE Pro will use the F/L setting in the element library for each beam member and multiple it by its length as projected in the chosen direction. So if we set the F/L value for the members as CF multiplied by the effective width dimension of the member, then a load for all the frame members can be developed all once for a unit pressure (p = 1.0). Then we can factor that load set by the pressure (p = 0.1 psi) to develop the frame component of load. This conservatively assumes all members throughout the entire tower and supporting structure (including focal support) are not shadowed by other members.

Approximate values for CF for a round member could be approximately 0.5, and for a rectangular member, a 1.2 factor is commonly used.

So, for the frame members (except those in the dish) we set up values of F/L = D*CF where D is the effective width dimension of the rectangular tube or the diameter of a round tube. The F/L values for the round members in the dish itself are set to zero since the dish will be loaded by another method.

<table>
<thead>
<tr>
<th>Description</th>
<th>F/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel Rect 2.5 x 1.5 x 0.1875</td>
<td>3</td>
</tr>
<tr>
<td>Polycarb Rect 3 x 2 x 0.25 x 0.25</td>
<td>3.6</td>
</tr>
<tr>
<td>Steel round tube 2.000 OD x 0.188</td>
<td>1</td>
</tr>
<tr>
<td>Steel round tube 1.750 OD x 0.219</td>
<td>0.725</td>
</tr>
<tr>
<td>AL 0.875 OD x 0.065</td>
<td>0</td>
</tr>
<tr>
<td>AL 1.000 OD x 0.083</td>
<td>0</td>
</tr>
</tbody>
</table>

Without selecting any elements or nodes, right-click and use **Set loads**.

In the dialog, open the **Projected** tab.

Set Z direction and Replace existing’. Check the item Use compensation since it will improve the distribution of loads in the tower somewhat.

Use the Apply projected load button. Note that Fz total load in the Z direction shown on the model metrics part of the dialog is 1130.7 pounds. This is for a unit pressure of 1.0. On the same dialog, open the File tab and use the Factor button. Enter 0.1 to factor the load for the design pressure of 0.1 psi, the value for Fz is now shown as 113.07 pounds. On the same tab of the dialog, use the Save button to save this load set with a name such as frameloads.lsb that you will recognize. You can exit and examine these frame loads on the structure.

**DISH LOAD**

Set the plate/beam selector to select plates. Wind over surfaces always create normal surface pressures so the wind in the Z direction sideways over the dish surface will create a normal load on the surface outward from the convex side. Assume that on the windward half of the disk (-Z) the pressure coefficient is -1.2 (acting outward from the curvature normal to the surface) and on the leeward (+Z) half of the disk the pressure coefficient is -0.8 (also acting outward and normal).

You can select all the plate elements on the windward side by using **Select/Elements/By slope**. Choose the XY plane and set; >=0 and <=90 to select the windward half of the disk. Use **Set loads** then open the Normal tab, REPLACE existing. Enter -1.2 in the intensity field for the windward part of the load. Press Apply normal load. Press OK to exit. This removed any previous loads (i.e. the frame loads) and applied loads normal to the dish only on the windward side.

You can select all the plate elements on the leeward side by using **Select/Elements/By slope**. Choose the XY plane and set >=90 and <=180. Use **Set loads** then open the Normal tab and check MERGE with existing (i.e. to keep the windward loads on the model as we add the leeward loads). Enter -0.8 in the intensity field for the windward part of the load. Press Apply normal load. Now, open the file tab and use the Factor button to factor the unit pressure load down to the design level pressure of 0.1 psi.

Before leaving the dialog, use Save button and save the dish load with a name like dishload.lsb. You now have both components of load saved and currently just the dish load is applied to the model. Go ahead and use the **Merge** button on the File tab and reload (i.e. merge) the previously saved frame load set (use a multiplying factor of 1.0). Now the model is fully loaded for wind in the Z direction on both the open frame members and on the dish. You could now use the **Save** button on the File tab of the loading dialog to save the full wind Z load set under a name of its own.
Use **OK** to exit the dialog with the model fully loaded with wind in the +Z global direction.

Solve the model. Use **Results/Maximum beam stress** to view the following results.

MAXIMUM/MINIMUM NORMAL STRESS, SN1 OR SN2 - MAXIMUM OF EITHER END

<table>
<thead>
<tr>
<th>Group name</th>
<th>Max/Min</th>
</tr>
</thead>
<tbody>
<tr>
<td>Antenna support</td>
<td>2616 -3037</td>
</tr>
<tr>
<td>Focal support</td>
<td>359 -359</td>
</tr>
<tr>
<td>Radial</td>
<td>21720 -27184</td>
</tr>
<tr>
<td>Ring</td>
<td>23768 -27476</td>
</tr>
<tr>
<td>Ring outer</td>
<td>1747 -707</td>
</tr>
<tr>
<td>Tower cross</td>
<td>1437 -1506</td>
</tr>
<tr>
<td>Tower post</td>
<td>6436 -6182</td>
</tr>
</tbody>
</table>

This is just as a result of wind alone in the Z direction, in reality it would be factored and combined with dead weight or other components of load which can be accomplished with the global loading methods described above.

**NOTES:**

1) **ASCE-7 methods:** The above method of determining frame loading is probably a little more conservative than it has to be since it does not account for downstream energy losses. There are other methods of handling tower loads per ASCE-7 by treating the tower shape (rectangular in this case) and the solidity of a side using formulas prescribed therein. If desired, one could calculate the total tower load by the ASCE-7 method then use the method described above but adjusted (factored with the 'Factor' button) to give the same total load as determined in ASCE-7. This results in a rational distribution of the ASCE-7 total load on the frame part of the tower.

2) **Direct frontal wind (Wind X in this case).** When determining wind in the X direction loading some reasonable values pressure coefficient for the dish would be

- 1.5 with the concave side facing windward, and
- 0.82 with the convex side facing windward.

These are estimated from aerodynamic handbooks. The framework may need to be adjusted in the X direction to account for shadowed elements.

3) **With this example,** all loaded plate nodes in the dish were also structural beam points so there were no issues associated with loads on unsupported non-structural plate elements. This is not always the case. With unsupported loads on nodes of non-structural surfaces, large deformation can take place distorting the results. In this case, once the loads have been applied you can use **Utilities/Special loading tools/Move plate loads to frame.** This will rationally redistribute any unsupported loads on plate nodes to adjacent frame boundaries.
**DOME2V.FEM (GEODESIC DOME)**

Geodesic domes are generally of two types; panel types or strut and hub types.

![Geodesic Dome Model](image)

**ICOSAHEDRON 2V DOME**

You can create these models with both panels and struts combined at the outset, but on rare occasions it is desirable to build a strut-only model first and add panels with hinged links to the struts and hubs to more precisely model the interface of panels, struts, and hardware. This is one way to do it.

**MAIN MODEL SET CONSTRUCTION**

This model is of a 2V icosahedron geodesic dome with a 30 foot (360 in) diameter base. This type is often used as a small residential greenhouse. This model was generated as a simple geodesic 'line' model (beams only).

**GEODESIC GENERATOR**

Radius = 180
Polyhedron type = Icosahedron
Frequency = 2
Class = I
Beams only
Geodesic fraction = Dome
   Dome base is unmodified
   Dome is cut at 0 on axis Z
Breakdown is by Method #1
Eccentricity on Z = 1.00
Zenith = Z
Check: *Use group names for like geometries*

The dome is created with group names already assigned for struts and certain nodes. Each different length strut has a name (Here just "B1" and "B2" since there are only 2 lengths in this dome) and the "Base" nodes have a name as well as the "Pole" node and "Center" node. Before developing the dome further, go to the Nodal Editor mode select nodes **by Group** and choose the group name "unnamed", then assign a group name "Hub" to them. That way, you can distinguish between actual hub nodes and the intermediate ones that will be created when the struts are subdivided in the next steps.

The property values for the "Main strut" are added to the Library and then assigned to all the dome beams.
Next the beams are selected all at once and assigned a 'reference length'. From the element editor mode pop-up menu and choose to set the reference length equal to their current length.

Finally all the dome elements are selected again and subdivided. We chose to discretize the struts into 6 segments for this model. You can go to the Nodal Editor mode, select 'by group' the newly created nodes with the default name "unnamed" and use **Hide selected** to simplify the view.

Save the main dome model.

**AUXILIARY MODELS**

Two additional 2D finite element models of the two different shapes of triangular panels associated with this type of dome were also made. There are only two different triangles, one is an isosceles triangle (call it Panel1) with sides B1, B1 and B2 and another equilateral with sides B2, B2, and B2. Find any Panel 1 triangle (you can see the group name B1 or B2 in the status bar for any selected beam element) and then select all the segments on the perimeter. Use **File/Save selected** and save that part as a new model called Panel1.

Also find any Panel 2 triangle, select its perimeter, and save it as well under the name "Panel 2". We found P1 and P2 below.

![Diagram of Panel 1 and Panel 2 triangles](image)

Load Panel 1. The auxiliary panel model is loaded into CADRE Pro by default in the same global orientation as on the main dome. To re-orient it to the XY plane necessary for construction, first select the 3 vertices (we selected in the order a, b, c as seen in the picture above) and use **View/Viewpoint/3-point-plane**. The action sets the current viewpoint to the precise XY planar viewpoint. But this is only a viewing perspective, to re-set the global system to align with this orientation use **Utilities/Constructs/Re-orient**. Repeat this process with Panel 2 as well.

The panels are constructed to look like the diagram below. Those two individual panel type models were made each, with the plate elements and frame connector extensions out to the triangular boundary of the original geodesic facet. They have side links set so that the links are oriented always in the same direction (in case we want to pin one end), here the outer ends of the links are always the axis node.

Group names are set up for each item in the individual panel models but elements are not assigned until actually merged into the model. The lines from the main dome onto which the panels are to be merged are given a property called 'main struts' in the main model, they already have group names B1, and B2 from the geodesic generator which are useful for determining where to merge the panels. The links next to the hub are given a different name (HL) in case we later want to make those elements part of the hub instead of a link like the others.
The group name nomenclature for the panel auxiliary models as shown below.

Once you have a properly oriented global system of the panel model in the XY plane, set up defined planes. First define one for the triangular face, then one normal to the triangular face parallel to the left side, then one normal to face parallel to the right side, and finally one normal to the face and parallel to the base side. Use these "Defined planes" and the 'Copy nodes relative' operation to locate construction nodes inward and normal to the edges (at 2.5 inches) to construct the inner and boundary which will become the 'Frame'. Then you can delete the outer boundary.

There are several ways to insert the plate elements. You can draw intermediate construction beams across as shown, use Join beams' to interconnect them and then use Utilities/Clad to apply the plate elements all at once. Then delete the intermediate construction beams.

Finally ensure adequate orientation nodes within the structure itself for all the elements. You can assign better ones to the frames later on. Delete all the defined planes, and all unused nodes. Check the model with Tools/Check model/Check for anomalies. You should have many 'invalid' properties since none are assigned, but there should not be any other issues.

Once the two types of 2D panel models are fully set up and saved, they can be merged onto the line model into an appropriate facet, then replicated around (at 72 degrees) with the clone utility. By NOT assigning properties to the panel models, you can select the panel after merging by selecting 'By property' and choosing the 'unassigned' elements. Then replicate. Then assign properties. This way each time a merge operation is completed the newly merged item is the only group of elements 'unassigned' making that set easily selected for replication. Accomplished judiciously, this amounts to 5 merging operations each followed by a replication operation, and an assignment operation to cover the full dome.

After the basic structure is created, many configurations of dome construction can be set up just by configuring the elements in the element library; some of them made to line elements (kept as place holders) and others assigned as substantial structure.

ELEMENTS AND PROPERTIES

Main struts are set as wood (2 x 6 nominal).
Hub links are set as 5 x 3/16 steel brackets
Frame struts are set as wood (1 x 4 nominal).
Side links are 1/2 inch steel bolts (Z pin at axis end)
Panels are polycarbonate sheet (1/2 inch thick)

LOADS

The model is loaded with snow (30 psf ground snow load) and dead weight.
Ground snow load: Pg at 30 psf = 0.208 psi
Roof snow load: Pf = 0.7*Ct*Cs*Pg
Where:

\[ \begin{align*}
C_t &= 1.2 \\
C_s &= 1.0 \quad \text{for } S \leq 45 \\
C_s &= 1 - \frac{(S-45)}{25} \\
C_s &= 0 \quad \text{for } X > 70
\end{align*} \]

The main dome struts and frames were given a 'reference' length equal to the hub to hub length even though they are divided into segments. After solution Select the main dome struts B1 and B2 examine the Results/Axial loads (use the 'Show selected' button to display just the selected items) and note that the \( P_{cry} \) and \( P_{crz} \) values (critical Euler load limits) are based on these ‘reference’ lengths rather than the segment lengths. These are the pinned-ended limit buckling loads for the struts without regard to any lateral support by the panels.

You can redefine the elements around the hub in any fashion to represent the actual hub hardware; including rendering some of them as virtual line elements if not needed.

**CONCLUSION**

In many, if not most cases, the panel to strut connection need not be modeled with this level of intricacy. Most can be modeled as simple frame structures with loads applied along the struts in a rational manner supported perhaps with a single panel set up and solved as a separate model to justify the panel integrity.

See the CADRE Analytic white paper #007 “Geodesic Dome Analysis” for a complete presentation of the typical complete dome analysis.

**DOME3V.FEM (Wood code checking and Shifting loads to frames)**

**CHECKING AGAINST THE WOOD DESIGN CODE**

This model is referenced in the User Manual as an example in checking against the National Design Specification for Wood.

Load the sample model, Dome3V.fem

The model is already loaded with a balanced snow condition.

Enable the wood code check feature using Options/Settings on the Beam tab.

Solve the model.

Shown below are data derived from the NDS specification for the No. 2 Fir 2 x 6 members.

\[ \text{Le} = 1.63(\text{Lu}) + 3*d \]

Where \( d = 5.5 \) inch for the deep dimension of the 2 x 6 struts

<table>
<thead>
<tr>
<th>Strut</th>
<th>Length</th>
<th>Le/Lu</th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>54.67</td>
<td>1.93</td>
</tr>
<tr>
<td>B2</td>
<td>63.28</td>
<td>1.89</td>
</tr>
<tr>
<td>B3</td>
<td>64.68</td>
<td>1.89</td>
</tr>
<tr>
<td>B5</td>
<td>66.84</td>
<td>1.88</td>
</tr>
</tbody>
</table>

One could use an approximate value of 1.90 and check all struts at once, or select each group name B1 through B5 one at a time and check each with the specific value of Le/Lu. If there is a wide variation the later approach would be more appropriate but in this case they are all close in bending slenderness.
The struts are oriented so that Iz is the deep dimension so Lez/Lu is the main value to set. You can always set the shallow dimension (Ley/Lu in this case) to zero.

The following values (in psi) and adjustment factors for flat use, $C_{FU}$, size $C$, and moisture $C_M$ are extracted from the NDS supplement "Design values for wood construction".

<table>
<thead>
<tr>
<th>Item</th>
<th>Published</th>
<th>$C_{FU}$</th>
<th>$C$</th>
<th>$C_M$</th>
<th>Enter</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_c$</td>
<td>1350</td>
<td>1.00</td>
<td>1.10</td>
<td>0.80</td>
<td>1188</td>
</tr>
<tr>
<td>$F_{by}$</td>
<td>900</td>
<td>1.15</td>
<td>1.30</td>
<td>0.85</td>
<td>1144</td>
</tr>
<tr>
<td>$F_{bz}$</td>
<td>900</td>
<td>1.00</td>
<td>1.30</td>
<td>0.85</td>
<td>995</td>
</tr>
<tr>
<td>$F_t$</td>
<td>575</td>
<td>1.00</td>
<td>1.30</td>
<td>1.00</td>
<td>748</td>
</tr>
<tr>
<td>$E$</td>
<td>1.60E+06</td>
<td>1.00</td>
<td>1.00</td>
<td>0.90</td>
<td>1440000</td>
</tr>
</tbody>
</table>

Go to the element library and make sure the $E$ modulus value is adjusted to 1.44E6 level to account for the moisture effect on the modulus.

Use Select/Elements/ByGroup and choose the struts with group names B1, B2, B3, B5.

Use Results/Code results/Wood beam stress

For a snow condition the duration factor $C_D$ would be 1.15. The struts can be assumed to be pinned at each end in all directions so set $K_y$, $K_z = 1$.

Press OK and in the next dialog, use the Custom button to uncheck the normal, principle and von Mises stresses leaving only the component stresses and length columns to be displayed along with the stress ratios.
Choose "Maximum of either end" for the location to read and evaluate stresses.

Press **OK** again and the data window displays showing only the selected data. This is a list of every single beam element (every segment of every member) along with its stresses and stress ratios evaluated under 6 different paragraphs of the NDS.

Scroll to the bottom and read $R_{\text{max}} = 0.1855$ and it is denoted as (T)3.9.1 indicating it is tension and derived from NDS Paragraph reference 3.9.1-1 concerning combined bending and tension.

<table>
<thead>
<tr>
<th>Fx/A</th>
<th>Fy/A*</th>
<th>Fz/A*</th>
<th>Mx/St</th>
<th>My/Sy</th>
<th>Mz/Sz</th>
<th>Length</th>
<th>Fc</th>
<th>3.8.1/3.7.1</th>
<th>3.3.1</th>
<th>3.9.1-1/3.9.2</th>
<th>3.9.1-2</th>
<th>R</th>
<th>Ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>94</td>
<td>5</td>
<td>0</td>
<td>20</td>
<td>19</td>
<td>87</td>
<td>66.84</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0.1855</td>
<td><a href="#">T3.9.1-1</a></td>
</tr>
<tr>
<td>-89</td>
<td>-5</td>
<td>0</td>
<td>-20</td>
<td>-32</td>
<td>-153</td>
<td>12.98</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td></td>
<td></td>
</tr>
<tr>
<td>-13</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-1</td>
<td>-19</td>
<td>61.48</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Press **OK** again and the data window displays showing only the selected data.

Scroll to the bottom and read $R_{\text{max}} = 0.1855$

There may be several of the same value or very close to it due to round-off. You can copy and paste a 0.185 value (in order to select all from 0.185 to 0.186) into the search box at the top of the data display and find one such value in the table. Note that one of them is P112*746*47 and it is from group B2. You can continue and find more of the same value, all from group B2. After closing the data display, use **Select/Elements/One by name**. Put the name P112*746*47 into the field and press **OK**. The critical element will be highlighted on the model. There should be 10 as shown in the picture below but you may not find all of them by the search due to round-off accuracy. These are from the major lower ring around the dome which typically carries tension and is often the critical tension member on this style of geodesic dome.
After closing the data display, use **Select/Elements/One by name**. Put the name P112*746*47 into the field and press OK. The critical element will be highlighted on the model.

The adjacent segment in the same member is also one with the same value as are the same pair at all 5 locations of symmetry on this style of geodesic under a symmetric balanced loading condition.

To display the stresses with the maximum stress ratio of each group or type of beam, you can use **Results/Code results/Wood beam stress maximum**.

You can also choose to show the length and name of one of the representative individual element having the maximum ratios.

<table>
<thead>
<tr>
<th>Group name</th>
<th>Axial</th>
<th>Shear-Y</th>
<th>Shear-Z</th>
<th>Torsion</th>
<th>Bend-Y</th>
<th>Bend-Z</th>
<th>Length</th>
<th>Name</th>
<th>Fc</th>
<th>R</th>
<th>Ref</th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>79</td>
<td>0</td>
<td>0</td>
<td>-4</td>
<td>-1</td>
<td>73</td>
<td>54.87</td>
<td>P133<em>795</em>47</td>
<td>1253</td>
<td>0.1551</td>
<td>(T)3.9.1-1</td>
</tr>
<tr>
<td>B2</td>
<td>94</td>
<td>0</td>
<td>0</td>
<td>-5</td>
<td>-1</td>
<td>87</td>
<td>63.29</td>
<td>P139<em>781</em>47</td>
<td>1220</td>
<td>0.1855</td>
<td>(T)3.9.1-1</td>
</tr>
<tr>
<td>B3</td>
<td>59</td>
<td>3</td>
<td>0</td>
<td>-6</td>
<td>-3</td>
<td>37</td>
<td>64.68</td>
<td>P1521<em>18</em>47</td>
<td>1212</td>
<td>0.1012</td>
<td>(T)3.9.1-1</td>
</tr>
<tr>
<td>B5</td>
<td>-73</td>
<td>1</td>
<td>0</td>
<td>6</td>
<td>3</td>
<td>-22</td>
<td>66.84</td>
<td>P36<em>2207</em>47</td>
<td>1199</td>
<td>0.061</td>
<td>(C)3.7.1</td>
</tr>
<tr>
<td>Base beams</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>5</td>
<td>-14</td>
<td>64.72</td>
<td>S264<em>960</em>48</td>
<td>1314</td>
<td>0.014</td>
<td>(T)3.9.1-1</td>
</tr>
</tbody>
</table>

**THE MAXIMUM STRESS RATIO SHOWN BY THIS METHOD IS 0.1855 FOR GROUP B2 AND ONE REPRESENTATIVE EXAMPLE OF A MEMBER AT THIS LEVEL IS P139*781*47 WHICH IS ONE OF THE 10 FOUND ABOVE WITH THE OTHER TABLE.**

**SHIFTING PANEL LOADS TO ADJACENT FRAMES**

This model is referred to in Help demonstrating the technique of shifting mid-panel loads to the frame.

This dome is loaded with a balanced snow load distributed to all dome nodes.

Check **Model/Gross properties** to see the resultant applied external forces:

Fx=0; Fy=0; Fz=-19061; Mx=587; My=0; Mz=0

The panels are necessary for supplying the load to the frame and used to build the load cases and often form an integral and essential part of the structural support. However, in some cases, one may want to check the frame alone under the same load case without considering the structural support of the panels. This is accomplished by simply setting the elastic modulus of the panels to a very low value so that they provide no support. But, in that case, loads
applied on mid-panel load result in enormous displacement and potential numerical instabilities along with poor results.

There is a CADRE Pro feature under *Utilities/Special loading tools* that allows one to redistribute rationally the existing load state from the mid-panel areas to the frame nodes only. It can be applied to the existing load on the model or even to a group of load sets saved in a folder.

To analyze the frame-only state, you would go to the element library and set the Elastic modulus of the panel member to a low number (e.g. 1.0). Then run the feature *Shift panel loads to frames*. Choose the option to work on the currently applied load set on the model.

This rationally redistributes the load to the frame members maintaining the same overall load. You can check the gross properties again and see that the total load and moments on the dome are still the same.

\[
Fx = 0; \ Fy = 0; \ Fz = -19061; \ Mx = 587; \ My = 0; \ Mz = 0
\]

If you turn on the load circles you can see that only nodes along the frame beams have a component of load. The stress in the frame can be much higher in this case.

This conservative analysis approach probably is not necessary for this particular configuration of dome but for thinner panels, fabric covers, or higher frequency geodesics where panel support is questionable, it may become essential. This feature can work on the applied loads on the model, or to a whole group of compatible load sets saved in a folder.

**DRUM.FEM (55 GALLON DRUM - FIXED CAPS)**

This is a standard 55 gallon steel drum that is supporting an internal pressure of 1 psi.

![55 gallon drum](image)

**DIMENSIONS**

55 gallon drums are typically Diameter =22.5, Height = 32 inches

**CONSTRUCTION**

1) Make the barrel from Quick Modeler cylinder template and save. (Use XZ plane, R = 11.25, L = 32, 16 sections, 32 sectors, do NOT check Iso triangles, plates only). For later convenience, select and assign a group name to all the barrel elements (e.g. "Barrel"). Save this component

2) Make a cap from Quick Modeler spheric template and save. (Use XY plane, R = 11.25, H=0, 6 rings, 8 sectors, 2 boundary adjusts, plates only) this makes a 32 support end cap that will match the end points of the barrel. For later convenience select and assign a group name to this component (e.g. "End cap"). Save this component.

3) Load the Barrel, then 'Merge' the end cap to drum barrel at the top end.

4) Select only the cap plates that were just attached and then 'clone' with Y=-32 to replicate a single copy of the cap at the bottom, end (checking 'Attach overlapping nodes' so that it will be attached).

5) Use the **Bandwidth Manager** to organize the model efficiently.

**ORIENTATION**

To make sure all elements are configured with consistent front/back (outside/inside in this case) orientation, go to the Element Editor, Select the entire model, and use **Orient: Orient to a point**. Choose an internal centerline node or enter a set of coordinates on the inside central region. This will switch any inconsistent plates to a consistent orientation.
ELEMENTS
The barrel portion is steel; E=2.9E7, V = 0.3, T = 0.88.
The end caps are steel: E=2.9E7, V = 0.3, T = 0.125.
Make library entries using Kirchhoff plate types and set these parameters.

GROUP NAMES
The barrel curved shell elements have a group name 'Barrel' and the end caps have a group name 'End cap'. These can easily be used to select and assign properties to the different portions of the barrel using Select/Elements/By group.

LOADS
Select the entire model (all elements).
Use Edit/Loads from the main menu bar or Right-click and use Set Loads.
Use the Normal load tab, positive, enter 1.0 (psi) in the intensity field, and Apply Normal Loads to the selected elements.
Notice that the panel says 770 nodes were loaded but the resultant value is Fx=0, Fy=0, Fz=0. That is because the internal pressure is balanced all around so there is no net resultant load. After leaving the Global load manager with OK. Turn on the force vectors temporarily to see the orientation of forces on nodes around the model. Ensure that they all point outward!

RERAINTS
One way to restrain (to stabilize the model) but have no binding influence is as follows:

Select the top cap center node and fix X and Z; all else FREE
Select the bottom cap center node and fix X and Z; all else FREE
Select the extreme front and back nodes on the middle of the barrel (level Y=16 at X=0) and for these two nodes FIX degrees X and Y; all else FREE

RESULTS
The nominal stress in the mid section of the barrel wall should approach the same result as classical stress analysis methods.

Classical average shell hoop stress = PD/2t = 127.84
Classical average tangential stress = PD/4t = 63.92
These are average values ignoring end effects (both bending and radial restraint effects of the end caps)

At the mid section far away from the end effects, this model gives.

Principal stress>
S1 = 63.60
S2 = 127.22
S12 = -31.81
Angle = 89.66 deg
Von Mises = 110.18

Near the ends there are higher compressive stresses on the outer surface due to the bending induced by the cap and due to the limited radial expansion resulting from the cap. See Drum1.fem to see how to hinge the cap to the barrel.
**DRUM1.FEM (55 GALLON DRUM - HINGED CAPS)**

This is the same model as *drum.fem* except that the caps are pinned rather than fixed to the barrel at the rim. Pinning plate intersections for this type of model is similar to that of the hinged plate exercise in Help.

CONSTRUCTION

Open CADRE Pro and open the file *Drum.fem* from the static sample files. This is a 55 gallon drum in which the end caps are integrally connected to the barrel so that moment is transmitted across the boundary.

Change this design so that the top end cap is of the hinged connection that does not transfer moment across the boundary.

The model is displayed with the boundary nodes and the cap center nodes visible and all other nodes hidden. The barrel and the caps have group names of *Barrel* and *End cap*.

Use the frame to select all the visible nodes on the upper cap (rim nodes and the center node). Then unselect the center node and immediately re-select it. This is to make the center node the last node selected.

Use *Select/Nodes/Re-order selection* or *(Ctrl J)*.

Choose to re-order by angle in a plane. In the dialog, choose the *XZ plane* (i.e. by angle about the Y axis). Notice that the dialog displays the center node (562) about which the angles are taken and used for sorting in order on the Y axis.

Press *OK* and the nodes are internally re-ordered as if you had selected them one-by-one around the perimeter of the cap.

The center node which was only needed for the re-ordering operation is now in the unselected state after this operation.
Next, with nodes still selected, use **Select/Elements/By group** and select the Barrel elements.

You could now install the hinge, but there is a trick that can be helpful. Any additional nodes selected that are NOT on the hinge line are ignored as the hinge is created, but the last one detected as not being on the hinge line is used as the guess for the reference node for the hinge beams.

So you can use this feature to advantage to set the reference node correctly at the outset. The top cap center node would be a good node for orienting the hinge beams.

So, re-select the top cap center node. Then, use **Utilities/Insert special structures/Surface hinge**.

Choose: **Closed loop**

A confirmation message is displayed prompting you to:

- Set proper a proper reference node,
- Evaluate the stiffness settings for the hinge beam,
- And run the bandwidth manager.

**VERIFY OR SET ORIENTATION**

If you select and check the orientation of any hinge beam element (right-click and use Show orientation) you will see that the reference node is at the center of the top cap. If not, you would need to select and set a proper orientation node for the hinge.

With orientation verified you can see that $I_y$ is the bending resistance of the hinge out of plane and $I_z$ is the resistance to in-plane bending.

**EVALUATE THE HINGE PROPERTIES**

Typically this hinge structure is just an artifice used to remove the tendency for moment to be transferred across a line in a surface. In some cases, one may need to adjust these values based on the actual loading conditions. For example if there were significant forces tending to cause shear along the hinge line, then the axial stiffness (Area) may need to be increased. Also, if the surface displaced enough to open a gap under load, then the bending stiffness ($I_y$ or $I_z$) may need to be increased in one or both directions.
Open the element library and find a new entry for *Hinge beam*. This is a pinned beam with very low axial stiffness, large bending stiffness, and zero torsional stiffness.

The initial artificial stiffness is set up based on a rod with diameter the size of the thickness of the selected plate. Then, it is adjusted to 10000 times stiffer in bending (Iy and Iz), and 1/10000 of the area (A). The torsional coefficient (J) is set to zero. The beam is fixed on the axis end (A-Ypin=1 and A-Zpin=1) and pinned on the origin end (O-Ypin=0 and O-Zpin=0). This is meant to be only an artifice that significantly resists displacement normal to the hinge line but allows free movement in torsion and only little resistance in axial displacement.

The size of the hinge pin (bending properties) must be carefully managed and perhaps adjusted so that it doesn't stiffen the plates in bending so much as to cause numerical issues, nor can it be so soft as to allow the plates to displace excessively relative to one another except in hinge rotation under the applied load condition.

The axial load must not be so stiff as to significantly resist the tendency of the plate to expand or contract along the hinge line if that is what the loading condition would tend to do. On the other hand, it should not be so soft as to allow significant differential motion of the left and right panels parallel to the hinge line.

If the hinge were an actual hinge with real structural properties that could be estimated then you may need to adjust the properties to match the actual hinge which may also involve adjusting the free pin ends to some partial pin fixity.

**SOLVE**

Solve and evaluate the hinge area to ensure there is no excessive opening of a gap in the hinge area. If so, you would need to increase the hinge beam bending stiffness by a magnitude or two. If there appeared to be some rotation of the cap you could increase the axial stiffness of the beam (but sparingly since axial stiffness can affect the hoop strength!).

The original model *Drum.fem* had a maximum von Mises stress on the top cap of 5051 psi. After installing the hinge and solving the model, the maximum von Mises stress is 9954 psi in the top cap.

Even with very large exaggerations of displacement there does not seem to be any separation along the hinge line so the stiffness settings appear to be adequate for this model.
HINGE LAYOUT

Normally the there is no gap between the cap and barrel and the beams are parallel to the hinge line. You can select the top cap and then use Utilities/Constructs/Translate. Set the Y value to 3 and do NOT check the detach option. This will lift the lid whose surface is not attached to the barrel except via the beam elements. Then you can see how the hinge beams are directed and situated around the perimeter.

ECCENTRICBEAMSTIFFENER.FEM (PANEL STIFFENERS)

BEAM STIFFENERS FOR PLATE PANELS

This model is the subject of "Getting started" exercise 23 in Help.

This illustrates the method of using eccentric beams (standard style) to stiffen sheets and panels made of plate elements. It is a simple example, analogous to a single strip out of a stiffened panel, so that comparison with theoretical can be easily done to build confidence in the method.

In this example, an aluminum 'T' beam is welded to a narrow strip of aluminum plate effectively making an I beam. Then, for final verification, a simple standard I beam model can be used to check the results to confirm this process.

![T section](image)

You can consider this configuration to be a narrow strip of a stiffened panel. This process of using eccentric beams to stiffen panels can be extended to wide panel models with multiple stiffeners in any configuration.

ELEMENT LIBRARY SPECIFICATIONS

There are two entries in the element library.

1) The strip of the panel is modeled with plate elements the same thickness as the flange.

![Stiffness properties and loading parameters](image)

2) The T beam is set up as a simple T beam using section properties derived from the Basic Shapes module.

In the Basic Shapes module, the information initially displayed for centroid location (C2) and section modulus (S2) is the larger offset to the centroid and consequently the smaller section modulus. These are usually the most important values for stress analyses. However, for this illustration, you will need the other distance from the centroid to the T flange outer face and the section modulus associated with that extreme fiber. You can opt to use these alternate values by checking the block for Use near edge modulus.
The information derived using the *Near edge modulus* is listed below.

- **T-section 3.25 x 6 x 0.25 x 0.25**
  - Area = 2.25
  - $I_Y$ (Inertia) = 4.50391
  - $I_Z$ (Inertia) = 1.89063
  - J (Torsion coefficient) = 4.7075E-02
  - $R_Y$ (Radius of gyration) = 1.41483
  - $R_Z$ (Radius of gyration) = 9.16667E-01
  - Ctr to edge on Y (i.e. $C_2$) = 6.66667E-01
  - Ctr to edge on Z (i.e. $C_1$) = 3.0E+00
  - $S_Y$ = 1.5013
  - $S_Z$ = 2.8359
  - $S_T$ = 0.0
  - Weight/length = 0.2205
  - $R_I$ = 0.0
  - $R_O$ = 0.0

The T beam offset must be the distance from T-beam's neutral axis to the beam attachment points at the plate flange (considered to be at the center of thickness $(3.25 - 0.6667 + 0.25/2) = 2.708$ inches. In this case it would be negative downward or -2.708 inches.

Set the T beam property in the Element Library as and Eccentric beam or the 'Standard' type.

Set the offsets to: $D_Y = -2.708$ and $D_Z = 0$

The plate elements are oriented so that the 'front' side is the underside which represents the outside face of the lower flange.
SOLUTION

Examining the origin node for the beam element just to the right of center with the eye view tool gives:

\[
\begin{align*}
    M_Z &= 3011 \text{ bending moment at centroid of the eccentric beam} \\
    F_X &= 3315 \text{ axial load at the centroid of the eccentric beam}
\end{align*}
\]

Stress on upper flange face is:

1) Calculating from section loads

\[
\text{Stress} = \frac{M_Z}{S_Z} + \frac{F_X}{A} = \frac{3011}{2.835} + \frac{3315}{2.25} = 2535 \text{ psi}
\]

2) Or read the stress directly with the eye view tool.

The plate flange stress is found by using Results/Plate stress maximums with plate stress S2, Vertex, Front side.

<table>
<thead>
<tr>
<th>Property name</th>
<th>Max/Min</th>
<th>Area</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flange plate</td>
<td>2500</td>
<td>2.25E+00</td>
<td>V56<em>55</em>153</td>
</tr>
<tr>
<td></td>
<td>109</td>
<td>2.25E+00</td>
<td>V4<em>37</em>136</td>
</tr>
</tbody>
</table>

The maximum stress is 2500 psi.

As expected, the stresses are the same (or nearly so) for the flange outside faces. That is because it is actually a symmetric beam.

You can also check the displacement at the center node and note that it is 0.109 inches.

THEORETICAL COMPARISON

The composite should be the same as the I-beam described as follows.

I-section 3.5 x 6 x 0.25 x 0.25

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area</td>
<td>3.75E+00</td>
</tr>
<tr>
<td>I_y (Inertia)</td>
<td>9.00391E+00</td>
</tr>
<tr>
<td>I_z (Inertia)</td>
<td>8.5E+00</td>
</tr>
<tr>
<td>J (Torsion coefficient)</td>
<td>7.93454E-02</td>
</tr>
<tr>
<td>C_y (outer fiber dist.)</td>
<td>3.0E+00</td>
</tr>
<tr>
<td>C_z (outer fiber dist.)</td>
<td>1.75E+00</td>
</tr>
<tr>
<td>S_y</td>
<td>3.0013E+00</td>
</tr>
<tr>
<td>S_z</td>
<td>4.85714E+00</td>
</tr>
<tr>
<td>R_i</td>
<td>0.0E+00</td>
</tr>
</tbody>
</table>
Ro = 0.0E+00
The theoretical displacement is
\[ PL^3/48EI = 0.10842 \text{ vs. } 0.109 \text{ given by our model} \]

The theoretical stress is
\[ M/S_2 = 12000/4.85714 = 2470 \text{ psi vs. the range of 2500 to 2535 given by our model.} \]

Furthermore, if we checked the range of S2 plate stress across the flange at the center rather than just the maximum we would find a range of 2400 to 2500 for the plate representation of the flange.

If one modeled the T portion of our model as a 'Standard' beam without using the eccentric offset, the total deflection would be -0.4854; almost 5 times greater!

See the sample file studwall.fem for a wall cladding on studs using eccentric beams.

**ELECTRICTOWER.FEM (Electric tower with combined loads)**

**DESCRIPTION**

The model is a typical power transmission tower constructed from standard aluminum round tubes. It illustrates building, setting, saving, and combining load sets using the load case assembler.

There are 5 types of tubes.
2.5 in. diam. at 0.049
2.75 in. diam. at 0.049
2.75 in. diam. at 0.058
3.00 in. diam. at 0.058
3.00 in. diam. at 0.063

These 5 items are set up in the Element Library along with their properties including the W/L dead weight parameter.

The applied loads include line tension, lateral wind (+X), and dead weight (-Y).

**WIND IN X DIRECTION**

Assume 90 mph or 132 feet/sec or 1584 inch/sec

\[ Q \text{ (psf)} = 0.00256*V^2 \] where V is in mph and the 'Area' is the side projected profile area.

\[ q \text{ (psi)} = 1.7778E-5*V^2 \]

\[ F \text{ (in lb.)} = q*CP*Area \] where the 'Area' is the side projected profile area in inches

Assume the effective pressure coefficient for these tubes (on the average) is \( CP = 0.9 \) and no other adjustment factors for exposure, altitude, or terrain are applied.

The wind pressure would be \( q = 0.1296 \text{ psi applied.} \)

It would be applied to the tube side profile area as projected on a plane normal to the wind.

The side profile area is the projected length multiplied by the diameter. The *projected load module* will take care of the projected length so we only need to enter a load per unit length as determined from the pressure and the tube diameter.
Each tube entry in the library is set up with F/L values set to:

\[
F/L = (\text{Diameter}) \times 0.1296 \text{ psi} >> \text{lb/in}
\]

- 2.5 OD 0.3240 lb/in
- 2.75 OD 0.3564 lb/in
- 3.0 OD 0.3888 lb/in

The wind load case is constructed by using ‘Set loads’ with nothing selected. Open the ‘Projected’ tab and set the X direction. Use the ‘Apply projected load’ button. This method uses the length of each beam as projected onto the ZY plane, and the assigned force per unit length (F/L) for the assigned member to calculate and distribute loads to the nodes of each beam in the model.

Resultant wind load in the X direction is:

\[
\begin{align*}
F_x &= 7.3404 \times 10^3 \\
F_y &= 0.0000 \\
F_z &= 0.0000
\end{align*}
\]

Open the File tab on the global loading dialog and save this load set for combining with other load conditions.

An alternative approach is to enter the just the diameter (or effective width) of the member for F/L in the library. Then when the load is applied it will be by default a 1.0 psi condition. Then open the File tab in the loading module and use the Factor button to factor the force by 0.1296 to get the actual pressure condition. Then save the load set. This makes it convenient to increase the diameters or widths as might be required for icing conditions, repairs, equipment, etc.

**DEAD WEIGHT**

You can stay in the global load dialog. Use the Remove button to remove the wind load, then open the 'Dead weight' tab.

Dead weight is weight of members only and is set using the W/L value set up for properties in the element library.

Note: If you were not already in the global loading dialog you would use Set loads with nothing selected and then open the 'Dead weight' tab.

Choose Y negative for the distribution of forces. Use the Apply dead weight button. The 'Dead weight' option calculates the force on each beam by using the full length of each beam and the assigned W/L parameter for each beam.

The resultant dead weight is:

\[
\begin{align*}
F_x &= 0.0000 \\
F_y &= -1.1094 \times 10^3 \\
F_z &= 0.0000
\end{align*}
\]

Open the File tab on the global loading dialog and save this load set for later merging.

**LINE LOADS (DISCRETE LOAD CASE)**

Could include wind and gravity but here are simply given as Fx=100, Fy=-100, Fz=375 and applied to each side of the tower on the arms connection points (80 and 88).

To make a discrete load set such as the line load, first ensure nothing is selected then use Set load. Open the ‘File tab’ and the Remove button to clean all loads from the model.

Then select the two line connection nodes (80 and 88) on the tower arm and use Set load.

Enter 100, -100, and 375 for the Fx, Fy, and Fz load for these points. Select OK.

The resultant load on the model for this pure line load case is:

Use Set load again with nothing selected, open the ‘File tab’, and Save the discrete line load set for later combining with other loads.

\[
\begin{align*}
F_x &= 200 \\
F_y &= -200 \\
F_z &= 750
\end{align*}
\]
COMBINED LOADS

Factored load combinations are required by many codes. For simplicity in this example, assume you want a factored combination for this model such as:

1.0 Dead + 1.0 Wind + 1.0 Line

Use Set load with nothing selected. Open the File tab.

GET AND MERGE METHOD

We could use the Get button to reload one of the load sets, for example Dead weight, apply a factor of 1.0, and load it. Then use the Merge button to load Wind with a factor of 1.0, and the Merge button again to load the Line loads with a factor of 1.0. Then the model would be fully loaded. This is an adequate method for a few cases like this.

ASSEMBLER METHOD

For several cases with many load sets and factors a more convenient method is to use the load assembler.

Use the Assemble button to go to the Assembler dialog, and then navigate to the folder where the load sets are saved.

All load sets in the folder will be displayed, set a factor of 0.0 for any set you want to exclude (or you can specifically select and remove them) and use the actual load factor (1.0 in this case) for those you want to include in the combination.

One can set up, and create many combined cases at one time using the Assembler dialog by adding more case columns. Here we are making just one case.

Click OK, and the combined load set is loaded on the model when returning to the Global load dialog. Use OK to leave the global loading module retaining the combined load set on the model.

The final resultant of the combined load can be checked under Model/Gross properties.

\[ F_x = 7.5404 \times 10^3 \]
\[ F_y = -1.3052 \times 10^3 \]
\[ F_z = 7.5000 \times 10^2 \]

SOLUTION:

Solution results in the following stress values for tube normal stress. This table is generated from Results/Beam stress maximums with "Normal stress" and "By property type".

MAXIMUM/MINIMUM NORMAL STRESS, SN1 OR SN2 - MAXIMUM OF EITHER END

<table>
<thead>
<tr>
<th>Property name</th>
<th>Max/Min</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.500 OD x 0.049</td>
<td>37336</td>
</tr>
<tr>
<td></td>
<td>-37199</td>
</tr>
</tbody>
</table>
EMBDBEAM.FEM (BEAM - FORCE DISPLACEMENT)

Model from Help ‘Getting started’ exercise number 15

This is an example of a beam without any known external loading but with a "specified displacement". The beam is 100 inches long embedded (fully fixed) in walls at each end.

The cross section is an I-beam 4 x 2 x 0.25 flanges and 0.1875 fillet radii. The properties are set up using the Basic Shapes module.

The midpoint is to be displaced downward by exactly 2.0 inches and you are to find the amount of load needed to accomplish this as well as all other internal loads in the beam.

See Help "Getting started"; exercise 15 for complete construction and configuration details.

VIEWING SUGGESTIONS

1) It is helpful in viewing this model to display the node names and also the node selection circles.
2) Turn on the load vectors.
3) Turn on the extents to help orient the viewer.

Additional viewing refinements to try:

4) Go to Options/Settings, open the Model tab and check the setting "Enable special node shapes" which will distinguish the different types of restraints used in the model.
5) Go to View/Nodes and check the item to Display load values and Include reaction vectors.
RESTRAINTS

Nodes 1 and 21 boundary restraints are set to 'fixed' in all 6 degrees of freedom (i.e. embedded in a wall).

Node 11 boundary restraints are set with all free except Y which is set as 'Displacement' and with a value of -2.0.

<table>
<thead>
<tr>
<th>Degree</th>
<th>Restraint type</th>
<th>One-way setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Free</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Y</td>
<td>Displacement</td>
<td></td>
<td>2.0</td>
</tr>
<tr>
<td>Z</td>
<td>Free</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Xrot</td>
<td>Free</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Yrot</td>
<td>Free</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Zrot</td>
<td>Free</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

SOLVING AND VIEWING

Solve using Static standard

Check **View/Nodes/Show load vector's** **AND Include reaction vectors** to display the reaction vectors.

Check **View/Nodes/Show load values** to see the reaction load values.

The reaction at node 11 is 50136 pounds pointed downward.

So, the required load to achieve this displacement (via node 11 reaction) would be -50136 pounds.

Better numerical display for the displayed reactions for this model is achieved by changing the number format to Fixed 0 (Options/Settings/General Tab).

CLASSICAL COMPARISON

\[ P = 192yEI/L^3 = 50136 \text{ pounds} \]  
(see Help: 'Reference formulas' section)

Another, more practical, example is the sample model on the same subject **SnapRing.fem**.

**EXPJOINT.HTM (HEAT EXCHANGER EXPANSION JOINT)**

The structure consists of a 32 inch pipe with 48 inch diameter expansion joint. Material is stainless AISI 301 steel at 0.125 thick.

![Diagram of expansion joint]

- The joint is known to contract by 0.25 inch due to the expansion movement applied to the joint from the expanding duct system beyond the joint. The joint itself is loaded with 2 psi internal pressure and is heated by a 200 degree temperature differential.

CONSTRUCTION

One can make these surfaces of revolution easily and directly in CADRE Pro using the **Utilities/Insert special structures** options along with the Clone (angular) and Clad features. See the model 'CoolTower.fem' and the Help "Getting Started" Exercise # 22 which is made in this fashion and presented in step by step detail.
BUILD THE SINGLE OUTLINE OF BEAM ELEMENTS

Start a new model File/New with extents (-15, -24, -24) (15, 24, 24) with a 1 inch grid. Turn on the snap tool and snap these node points onto the screen.

A -15 16 0  
B -5 16 0  
C -5 18 0  
D -3 18 0  
E -3 22 0  
F -1 22 0  
G -1 24 0  
H 1 24 0  
I 1 22 0  
J 3 22 0  
K 3 18 0  
L 5 18 0  
M 5 16 0  
N 15 16 0  
P 0 0 0 (for reference)

The letters above are just for reference in following this note, the nodes can be any name you want. In this model they were actually numbered 1 to 15. In fact, you can copy the above lines, paste to a single cell of a spread sheet, then select and copy just the 3 numbered columns from the spread sheet and paste to "Notepad". Then save it as a plane text file such as 'Points.txt'. Then you can 'Import all the points at once when in the Nodal Editor mode using the Import button on the Nodal Editor panel. To learn more about this process see Help Import nodes

The single line of beams was first constructed by first setting 14 points. Lines are drawn between points AB; ED; GH; JK; MN (use point P for reference).

Then arcs are drawn using center points C, F, I, L using Utilities/Insert structures/Arcs and circles (setting here was for 4 segments on a 2 inch radius).

The flat lines AB and MN were then subdivided to 2 inch divisions and the other straight lines into 1 inch divisions creating a segmented single line from one end to the other.

Notice that the segments in the radii are kept short to provide finer meshing at these points. This is one of the advantages of building up the models directly in CADRE Pro. You can carefully control and organize the meshing for precision rather than accepting a random and sometimes awkward meshing provided by CAD applications.

CREATING THE WIRE FRAME

In the Element Editor mode, select all the elements and assign node P as the orientation node for all of them. Then, you can delete the 4 radii center point nodes which are now unused but keep centerline node at P.

Both beams AND nodes are selected simultaneously and the Utilities/Clone method is used to replicate the line into a surface of revolution about the centerline with cross connections at the selected nodes (the choice here was 7.5 degrees between each replicate (48 copies)). Check the option to connect all overlapping nodes. Use the X axis of rotation and point P for the location of that axis.
That constructs the wire frame of revolution.

CLADDING

There will be duplicate beam elements along the closing seam. Use the Tools/Check model and repair until the duplicates and collocated node anomalies are eliminated. Orientation and property assignments don't matter at this point since the entire wire frame will eventually be removed.

Use the Utilities/Clad with the 'quad' option to install plate elements. Use point P to orient the plates so that the front side is outside and the back side is inside (Hint: select the central point P first).

The wire frame is only a construction artifice. So now you can select all the beam elements that formed the wire frame and delete them.

One strip of plates will have a oddly oriented diagonal. If desired, it can be reversed by selecting them all (pair by pair) and then using (Tools/Switch plates diagonal).

RERAINTS

Fixed restraints are on the left rim while the right rim is given a "specified displacement" type of restraint of -0.25 inch representing the expansion movement externally applied to the joint from the duct system beyond the joint.

One must be careful to apply restraints in a fashion not to bind the joint in the radial direction yet provide for rigid body stability. In this regard the 4 end nodes on the XZ plane are also restrained in Y preventing vertical movement. Then two nodes, one on each end (opposite) are also restrained in Z to prevent movement in and out.

PROPERTY AND TEMPERATURE

Properties are set up as AISI 301 stainless steel at 0.125 thick. With the Element Editor mode, the entire duct is selected for property assignment using the Property button.

THEN RE-SELECT ALL THE ELEMENTS AGAIN FOR TEMPERATURE ASSIGNMENT USING THE PRELOAD BUTTON. HERE WE USED THE MATSAMPPF.TXT FILE TO IMPORT THE EXPANSION COEFFICIENT AND SET THE TEMPERATURE CHANGE TO POSITIVE 200 DEGREES (+ IS WARMER).

LOAD

Then select the entire model again, right-click and use Set loads from the pop-up menu. Under the Normal tab set up a 2.0 psi pressure condition.

RESULTS

VON MISES STRESS OUTSIDE (PLATE FRONT SIDES) AND INSIDE (PLATE BACK SIDES)

<table>
<thead>
<tr>
<th>Side</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>FRONT SIDE</td>
<td>2.560E+04</td>
<td>PSI</td>
</tr>
<tr>
<td>BACK SIDE</td>
<td>2.398E+04</td>
<td>PSI</td>
</tr>
</tbody>
</table>
**FLANGETUBE.FEM (Tube with flange model)**

A flange and tube assembly must support a 100 pound lateral handling force applied on the outer 2 inches of the tube. The 0.25 inch thick flange and 0.125 inch thick tube are 6061-T6. The flange plate is pinned around its perimeter and sets on a hard surface.

This example illustrates:

1) Hole model template
2) Cylinder template of the Quick Modeler
3) Merge utility
4) One way reactions

The model components are created separately and merged together with the Merge feature.

Search Help with "Hole model utility" where the construction of this specific tube and flange model is presented with a step by step example.

**CONSTRUCTION**

**HOLE MESH UTILITY**
- Center point = 0,0
- Upper right point = 3,3
- Vertical divisions = 3
- Horizontal divisions = 3
- Radial divisions = 6
- Hole radius = 1
- Hole segment count = 24
- Aspect ratio (Maximum) = 1.911
- Discretization = Log
- Quad = 4
- System = NONE
- Uniform nodal distribution around the hole

**QUICK MODELER - CYLINDER TEMPLATE**
- Center point = 0, 0
- Plane = XY
- Plates only
- Radius = 1.0
- Length = 4
- Sectors = 24 (to match the hole segment count!)
- Sections = 16
LOADS
The handling load is applied by selecting the plates on the outer 2 inches of the tube on the -X side only.

Then use, **Set loads**. Open the Projected tab. Set: Positive X. Enter an intensity of 25 psi.

The projected cross section is 2 x 2 or 4 square inches at 25 psi will produce 100 lb as projected on the tube centerline plane.

RESTRAINTS
All flange outer edge nodes are pinned.

All other flange nodes as well as the rim around the hole are set up as one-way nodes allowing them to lift, but not to sink into the hard surface. The positive option is selected to indicate that the restraint is fixed only when the reaction would be positive (i.e. holding the model up). If the reaction force would be negative, the restraint is removed and the solution recalculated.

The feature results in a multiple converging solution which takes some time.

SOLUTION
The maximum von Mises stress is about 6500 psi in the flange end of the tube at the seam. One would have to consider the weld allowable and weld material.

WELD LINE ANALYSIS
The stresses around the hole perimeter can be used to determine the normal and parallel running loads on a weld line provided that the stresses can be determined in a direction normal and parallel (or tangent) to the weld line.

The plate elements abutting on the hole rim have a local coordinate X, Y system as long as no other regional system is applied and this provides normal and parallel stresses. In this regard, X should be parallel to the hole and the Y normal to the hole.

In general, these may not always be properly oriented or in the correct direction but you can always reset the local system in a plate element by selecting it, using **Show orientation**, then while orientation is shown on the plate, using **Rotate local system** from the pop-up menu.
Use it repeatedly until properly oriented. You should ensure that all of the plate elements with a side abutting the weld are oriented in this fashion if you want to determine running loads around the hole and even plot them vs. distance around the perimeter.

This should be accomplished on the abutting plates of both the flange and tube.

When properly set up these stresses, Sx and Sy, could be used, along with plate thickness, to calculate the running loads on a weld. Then those loads may be used with the weld fillet dimensions to find stresses in the weld.

**FLATPLT.FEM (RECTANGULAR DIAPHRAGM)**

This is the simply supported plate diaphragm model used in Help ‘Getting Started’ Exercise 8.

```
  10  P = 1.0 psi
  ss

  20 inches
  ss
```

The edge nodes have a Group Name in order to make it easy to select them and check different boundary conditions.

Each plate element in the diaphragm is also assigned a 'regional coordinate system' in X and Y so that all aligned stresses, Sx, Sy are oriented the same way irrespective of the local orientation of each plate. To view this system, select any plate, right-click, and choose 'Show orientation'. The system data setup can be view and edited by Edit/Defined system.

See the Help tutorial “Getting started”, Exercise 8.

**FLYWHEEL.FEM (Rotational loads)**

This is the model from the Help tutorial, Exercise 29

0.25 lb mass 18 pcs

1.0 in.

Aluminium T=0.25 in.
This exercise involves a spinning aluminum disk with concentrated mass points on the rim. The disk is 10.5 inches in diameter and turns on a 1.0 inch shaft. There are 16 evenly spaced concentrated mass nodes (0.25 lb mass) on the rim in addition to the mass of the disk itself.

The requirement is to solve for the stress in the disk under 3 load conditions:

Case 1: Steady rotation at 2000 RPM
Case 2: Acceleration from zero rotation at 2100 radian/sec²
Case 3: Braking from 2000 RPM at the rate of 2100 radian/sec²

The main objective of the exercise is to practice using the global rotational loading and discrete rotational loading features and assembling combinations of angular and centrifugal forces to resolve realistic stress states. The model for this exercise is purposely kept very simple and easy to construct in order to emphasize the main objective which is distributing the rotational loads over the model. As presented in the sample files, the model is loaded with Case 3.

See Help "Getting started" Exercise 29 for complete details.

**FRAME.FEM (QUONSET FRAME)**

This is the frame used to construct the Quonset hut in Exercise 10 of the Help "getting started" exercises. The exercise also describes how to make this frame model from a saved list of nodal coordinates.

**IMPORTING NODES**

The coordinates must be given in a list of X, Y, Z with each separated by a Tab character as shown below.

```
-600.00  0.0000  0.00
-581.94  80.270  0.00
-503.13  227.58  0.00
-387.89  348.56  0.00
-244.57  434.44  0.00
-83.540  478.98  0.00
-147.25  395.96  0.00
-283.84  337.43  0.00
-399.87  244.61  0.00
-504.82  124.20  0.00
 0.0000  0.00
 0.00478.98  0.00
 0.00415.95  0.00
```

Or, using the comma for a decimal separator:

```
-600.00  0.0000  0.00
-581.94  80.270  0.00
-503.13  227.58  0.00
-387.89  348.56  0.00
-244.57  434.44  0.00
-83.540  478.98  0.00
-147.25  395.96  0.00
-283.84  337.43  0.00
-399.87  244.61  0.00
-504.82  124.20  0.00
 0.0000  0.00
 0.00478.98  0.00
 0.00415.95  0.00
```
The decimal marker with "," or "." will be interpreted correctly either way but must be consistent throughout the list and
the list must not contain both points "," and commas ",," in any fashion.

Select and save the appropriate list above to a simple text file perhaps called Nodes.txt.

Start a new model: Extents -600, 500, 0 by 600, 0, 0 with grid 24.

Use the Import button on the nodal editor panel. Choose the file Nodes.txt.

After importing and drawing elements the frame would look like this.

MAKING THE FULL FRAME

First add the beams to the nodes, assign group names to the inner chords, outer chords, and diagonals. Then set up
the library with the section properties for chords and diagonals.

Then, the brief steps for completing the full truss are as follows.

1) Use Utilities/Mirror.

2) Choose to mirror (reflect) along the X axis and to locate the reversed image on the "Positive" X side of the original.

3) Press OK and you have the complete frame.

Note: You could perform this operation with the Clone command to copy the original to the right at 600 units, then
select the new copy and use the Flip operation to reverse it, and then use the Attach nodes operation to connect it
fully. The Mirror command performs these three operations behind the scenes.

Although perfectly fine as is, assume you wanted the top horizontal element at the top of the frame to be continuous
and undivided. You could just delete both parts of it and redraw one across or just delete one part and drag the node
(#12) to the opposite side and drop it (then select and use Attach nodes).

RENAME NODES

Now rename the nodes in a neat fashion:

8) Go to the Nodal Editor mode. Select all the nodes

9) Right-click and use Rename nodes

10) Set Renumber start at 1, increment by 1; check 'By increasing X'; check 'Pad left with' character "0". Press OK.

11) You can select and rename the ground node to #22

The result should be the full frame fully connected.

You can also assign special user names to the elements in exactly the same manner as was accomplished for nodes.
The only differences are:

1) That you must enable this function in Settings/General tab and checking Enable element user names.
2) And you must select any or all the elements and use *View/Elements/Show names* in order to display the user name on the screen for individual elements. This creates a lot of clutter so element names are usually hidden and displayed only when investigating specific elements.

This sample model has an additional unassigned property added to the library which is eventually used for the interconnection of lateral beams between the multiple frame trusses while constructing the full model of a Quonset hut.

Additional models *Quonset.fem* and *Quonset1.fem* have further information.

See Exercise 10 in Help “Getting started” for complete details of building, loading, and analyzing the Quonset huts.

**FRAMELOADING.FEM (FRAME LOADING EXERCISE)**

This is a bare frame model that is the subject of the loading exercise Help "Getting started", Exercise 26 and in the CADRE Analytic white paper #006. The white papers are available at [https://www.cadreanalytic.com](https://www.cadreanalytic.com) for evaluators as well as licensed users.

Exercise 26 in help includes developing unit load sets, primary load sets, and finally complete combination load sets for a finite element model.

The white paper covers construction details and 2 methods of loading the frame.

- Directly on the frame members using define planes and unit loads
- Applying cladding and loading the cladding with unit loads

Exercise 26 in help is about the second process where artificial cladding in installed. The loads are developed as if the structure had walls resisting the external pressure but only the frame is considered to resist the loads.

Open frame structures in which there are no pressure resisting walls (e.g. open lattice towers) where wind can blow through the structure are loaded in an entirely different manner (See: ‘Comtower.fem’).

This unloaded frame version is provided for those who wish to perform the external loading exercises without first having to construct the model. Defined regional systems for the roof slopes and the ground plane are already set up on the frame. Use *Edit/Defined systems* to select and view them (RoofSlope1, RoofSlope2, and Ground plane)

The defined systems are not needed for the cladding method of exercise 26 in Help.
**FUSELATE.FEM (FUSELAGE MODEL)**

This is a truss, roughly in the form of an airplane fuselage structure.

It is not designed to represent an actual airplane and is only used here to illustrate some of the techniques of restraining such a model that is, by nature, unrestrained and transferring concentrated load masses to portions to the main structure.

Typically the coordinate system for an airplane would use Z upward, X forward, and Y lateral. For presentation convenience, this model is set up with X forward, Z lateral, and Y upward so that it is immediately shown as it would appear to a viewer looking from the side. Any system can be used.

The chosen consistent units are the inch-pound system.

The load condition represents a balanced 3.8g positive maneuver (pull up).

**WEIGHTS**

- **W-eng** = Engine & prop weight 200 lb concentrated at node E
- **W-else** = Payload, equip, and all else = 750 lb concentrated at node P
- **W-fuse** = Fuselage structural weight = 267.3 lb distributed over all tube structure nodes
- **W-tail** = Weight of empennage = 75 lb distributed over 3 tail nodes
- **W-wing** = Weight of wing = 200 lb divided between nodes R1 and R3 (fwd. wing spar attachment)
- **W-fuel** = Weight of fuel = 200 lb divided between 4 nodes R1, R2, R3, R4

**GROSS LOADS**

- Total static load is 1692.27 lb
- \( L_{\text{tail}} \) = lift on the tail in the maneuver = 100 lb distributed to the 3 tail nodes
- Total load on the aircraft in the maneuver is \( 3.8 \times W_{\text{static}} + L_{\text{tail}} = -6430.63 - 100 = -6530.63 \)

**LOADS AT CONCENTRATED CENTERS**

Centers of concentrated load points are connected with 'dummy' load transfer rods. They are 'relatively' rigid with zero weight and pinned at both ends and used only to get a center concentrated load into hard points of the structure.

**BOUNDARY CONDITIONS**

- Points R1 and R3 represent the front spar attachment and are pinned (i.e. Xrot, Yrot, Zrot all free; X, Y fixed allowing pitch about the Z axis. Z is fixed on R1 to prevent lateral movement of the structure.
- Points R2 and R4 represent the rear wing attachment points are free except that Y is fixed to react the tendency for the fuselage to pitch.

**BUILDING THE LOAD CONDITION**

The weight definitions are each set up and saved separately as individual load sets. Then, using the global load manager, they are reloaded and merged together using the 3.8g load factor on all weight sets and a factor of 1.0 on the tail load set. (i.e. Get \( L_{\text{tail}} \) using 1.0 factor, Merge \( W_{\text{eng}} \) using n factor, Merge \( W_{\text{fuse}} \) using n factor, Merge \( W_{\text{wind}} \) using n factor, Merge \( W_{\text{fuel}} \) using n factor, Merge \( W_{\text{else}} \) using n factor. The resulting total lifting load of 6530.63 is then properly distributed over the structure.
BALANCING CONDITION

If this were actually a “balanced” maneuver condition at 3.8g then the -100 pound tail load would be the required balancing tail load. In this case, the differential moment between front and rear spar must be the same as the aerodynamic pitching moment of the airplane (assuming the aerodynamic center is at the front spar). The reaction load on the rear wing attachment is 609.7 pounds on each side.

\[ M = 2 \times 12.07 \times 609.7 = 14718 \text{ in lb (nose down moment)} \]

**GARAGE.FEM (Wood frame wind and snow loading)**

**WOOD FRAME MODEL SET UP FOR WIND AND SNOW LOAD ANALYSIS**

**CONSTRUCTION**

The left side wall was created first. The side walls are 2x4 studs spaced 16 inches with a 4x4 top cap and base sill. It is constructed from the Quick modeler initially as a 'Plate mesh 1' model with 18 by 9 mesh with 288 x 96 dimensions. Then a 18 by 9 wire mesh is superimposed over it using the 'Insert mesh' utility. It is then completely configured with assignments of properties and orientation including the window configuration.

The single wall is replicated at 144 inches to create the right wall. An end wall is created using the 'Insert mesh' utility with 9 by 9 mesh, once with 'Plate mesh 1' then again with 'Wire mesh 1' to overlay frame on the cladding.

A ridge point is created at the 126 inch high apex above the end wall and the gable is drawn and clad as shown. Only the dark lines are the actual struts within the truss, the white lines are plate boundaries for which there is no truss strut.

This configuration allows the roof slope to be divided into 6 sections along the slope for better load distribution even though we only have laterals at two locations along the slope.

The total end wall including gable is replicated at the far end.

Finally, ONLY the truss beams of the gable (dark lines in the picture including the boundary) are selected and replicated 17 times at 16 inches along the Z axis to install the roof truss frames.

The roof slope is created in the same manner as the end walls. Select the 4 corners and insert a 18 x 6 mesh of 'Plate mesh 1' then again with 'Wire mesh 1'. Repeat the operation on the opposite side.

There are many duplicate elements created in the above process at the boundaries so be sure to clean up the model and make sure all nodes are attached.

Members are best set up assigned and oriented during the build up process before replication but some work is still needed to prepare ends that probably should be pinned and reoriented. Studs are oriented with the stiffest axis normal to the wall as are the truss rafters. The roof laterals are oriented flatwise as roofing attachment boards. Studs are pinned at the ends in the lateral direction. Horizontal members in the walls are assigned a 'line' property since they do not exist in the real structure and only served to discretize the wall during the initial process (they could be deleted as well). Refinements such as pin or partial pin joints and orientations are made typically based on study of actual design drawings or the actual structure, if available.
Finally, it is very important to orient all surface plate elements for a proper inside/outside orientation for the loading operations to work properly. Create a single node in the floor center and use it to orient all plate surfaces to that point for proper front/back orientation.

Open the Element Editor mode and select the center node on the bottom of the floor. Select all plate elements and use the Orient button on the element editor panel. Use ’Orient to a point’. In the dialog just choose OK since the preselected point is already set up.

DEFINE SURFACES FOR LOADING

The actual mean height is \( h=111 \) inches but we are using a slightly more conservative value of \( h=112 \) which lines up precisely with dimensions on the structure.

The roof is divided into 3 stages parallel to the ridge, 2 of them at 112 inches, each representing approximately one mean height length and the remainder on the downwind side at 64 inches.

The 10 wind surfaces used for loading the model are shown in the following diagram. This is to correspond to the required stages in ASCE-7 for wind parallel to the ridge from the front wall. The L and R in the diagram are for Left and Right sides.

WIND PRESSURE COEFFICIENTS

Using ASCE 7-10 along with a gust factor \( G \) of 0.85 the following factors are found for each surface on the model. The 10 surface names correspond to the group names assigned to various surfaces which are strategically named to facilitate applying wind loads both in the direction normal and parallel to the ridge as well as internal pressures.

<table>
<thead>
<tr>
<th>Surface</th>
<th>Ridge Normal (+X)</th>
<th>Ridge Parallel (-Z)</th>
<th>Internal (outward)</th>
<th>Internal (inward)</th>
</tr>
</thead>
<tbody>
<tr>
<td>RL1</td>
<td>-0.41905</td>
<td>-0.765</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>RR1</td>
<td>-0.51</td>
<td>-0.765</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>RL2</td>
<td>-0.41905</td>
<td>-0.425</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>RR2</td>
<td>-0.51</td>
<td>-0.425</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>RL3</td>
<td>-0.41905</td>
<td>-0.255</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>RR3</td>
<td>-0.51</td>
<td>-0.255</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>Wall left</td>
<td>0.68</td>
<td>-0.595</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>Wall Right</td>
<td>-0.425</td>
<td>-0.595</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>Wall Front</td>
<td>-0.595</td>
<td>0.68</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>Wall Rear</td>
<td>-0.595</td>
<td>-0.425</td>
<td>-0.18</td>
<td>0.18</td>
</tr>
</tbody>
</table>

CREATE "NORMAL" UNIT LOAD SETS

Wind loads are always normal to a surface and include all external surfaces. The 10 unit load normal load sets are created all in one operation. Select nothing and use Set load. Open the Normal tab. Set Negative, Plates only, and Use compensation. Press the Unit sets button. Choose or create a folder to place the unit load sets (preferably an empty one). These load sets correspond to a 1 psi condition on all surfaces oriented positive inward as the dialog is
set up. The Negative choice is necessary to make the positive pressure point inward corresponding to the sign convention of ASCE-7.

ASSEMBLE THE WIND LOAD COMPONENTS

A wind load component is created for each column of the above table. Open the File tab in global loading dialog. Use the Assemble button. Navigate to the unit load folder. Add 3 more columns to the table so that there are 4 components. You can rename the columns (cases) to WindX, WindZ, Int-Inward, and Int-Outward corresponding to the 4 components for which we have pressure coefficient distributions in the above table.

Fill in the wind pressure coefficient values.

![Load set assembler](image)

When finished, right-click on the header and use the Save configuration feature to save the layout in case you need to return and correct or repeat the operation.

You can save all 4 component cases to a separate folder. Exit and check the cases.

You can use Model/Resultant loads and generate a global total load table.

<table>
<thead>
<tr>
<th>Load set</th>
<th>Fx</th>
<th>Fy</th>
<th>Fz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Int-Inward</td>
<td>0</td>
<td>-7465</td>
<td>0</td>
</tr>
<tr>
<td>Int-Outward</td>
<td>0</td>
<td>7465</td>
<td>0</td>
</tr>
<tr>
<td>WindX</td>
<td>31337</td>
<td>19265</td>
<td>0</td>
</tr>
<tr>
<td>WindZ</td>
<td>0</td>
<td>21542</td>
<td>-17662</td>
</tr>
</tbody>
</table>

Notice that internal pressure components only have a lift value, Fy, but are otherwise zero.

NET WIND CASES FACTORED TO WIND PRESSURE

Assume the design wind condition is 130 miles per hour.

Using a height factor, Kh = 0.85 and a directional factor, Kd = 0.85 then
Wind pressure \( (Q_h) = 31.25 \text{ psf or 0.21707 psi} \)

The external wind case must be combined with internal pressure case and adjusted for actual wind pressure. Here for brevity we only used the outward acting internal pressure but in general all combinations must be determined.

Go back to the global load module with Set loads and open the File tab.

Use the **Assembler** button. Navigate to the folder where the four wind components are saved. Add 3 more columns to the table. Rename the columns and configure the table as shown below.

<table>
<thead>
<tr>
<th>Load set</th>
<th>WXin</th>
<th>WZin</th>
<th>WXout</th>
<th>WZout</th>
</tr>
</thead>
<tbody>
<tr>
<td>Int-Inward. lbsb</td>
<td>0.21707</td>
<td>0.21707</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>Int-Outward. lbsb</td>
<td>0.0</td>
<td>0.0</td>
<td>0.21707</td>
<td>0.21707</td>
</tr>
<tr>
<td>WindX. lbsb</td>
<td>0.21707</td>
<td>0.0</td>
<td>0.21707</td>
<td>0.21707</td>
</tr>
<tr>
<td>WindZ. lbsb</td>
<td>0.0</td>
<td>0.21707</td>
<td>0.0</td>
<td>0.21707</td>
</tr>
</tbody>
</table>

Save the configuration in case you want to return and revise it.

This will create the 4 wind cases as named in the column headers.

Alternatively you can simply use the **Get** and **Merge** buttons as follows:

Use the **Get** button to load the WindX component (factor of 1.0)

Use the **Merge** button and merge the Int-Outward pressure component (with a factor of 1.0).

Next use the **Factor** button and enter a factor of 0.21707 psi adjust down to the actual wind pressure.

Save this load set to another folder for the **Wind load cases** as WXout

Repeat using the WindZ component and save it as WZout

Use the **Get** button to load the WindX component (factor of 1.0)

Use the **Merge** button and merge the Int-Inward pressure component (with a factor of 1.0).

Next use the **Factor** button and enter a factor of 0.21707 psi adjust down to the actual wind pressure.

Save this load set to **Wind load cases** as WXin

Repeat using the WindZ component and save it as WZin

Use **OK** to exit the global loading module.

Use **Model/Resultant loads** to make a table of net wind loads for these four cases.

<table>
<thead>
<tr>
<th>Load set</th>
<th>Fx</th>
<th>Fy</th>
<th>Fz</th>
</tr>
</thead>
<tbody>
<tr>
<td>WXin</td>
<td>6802</td>
<td>2561</td>
<td>0</td>
</tr>
<tr>
<td>WXout</td>
<td>6802</td>
<td>5802</td>
<td>0</td>
</tr>
<tr>
<td>WZin</td>
<td>0</td>
<td>3056</td>
<td>-3834</td>
</tr>
<tr>
<td>WZout</td>
<td>0</td>
<td>6297</td>
<td>-3834</td>
</tr>
</tbody>
</table>

These are some of the basic wind loads. They would be further factored and combined with dead weight, snow loads, etc to create the design load combinations for analysis.

**SNOW LOAD PRESSURE**

Assume a design ground snow load condition is \( P_g = 40 \text{ psf or 0.27777 psi} \)

Assume the importance factor, \( I_s = 1.0 \); exposure factor, \( C_e = 1.0 \); thermal factor, \( C_t = 1.0 \)

The flat roof snow load, \( P_f = 0.7C_eC_tI_sP_g = 28 \text{ psf = 0.194444 psi} \)

The actual snow pressure, \( P_s \), is corrected by slope a slope factor, \( C_s \), but \( C_s = 1.0 \) for this small slope of 22.26 degrees

\[ P_s = C_sP_f = 0.194444 \text{ psi} \]

The balanced snow condition will have \( P_s \) applied evenly all over both slopes of the roof and projected normal to the ground plane.
Consider unbalanced snow with wind from the left (i.e. the WindX case above). The unbalanced snow condition for this particular gabled roof would have zero snow load on the windward side (left) but the full value of the ground snow load $P_g$ applied to the downwind (right) side.

You could rename the roof to have only two regions, left and right, but it isn't necessary, just treat the 3 regions on the left all the same and the regions on the right all the same as you use the assembler to create the unbalanced case.

Snow pressure $P_s$ for both balanced, $S_b$, and unbalanced, $S_u$, snow conditions.

<table>
<thead>
<tr>
<th>Surface</th>
<th>$S_b$</th>
<th>$S_u$</th>
</tr>
</thead>
<tbody>
<tr>
<td>RL1</td>
<td>0.194444</td>
<td>0</td>
</tr>
<tr>
<td>RR1</td>
<td>0.194444</td>
<td>0</td>
</tr>
<tr>
<td>RL2</td>
<td>0.194444</td>
<td>0</td>
</tr>
<tr>
<td>RR2</td>
<td>0.194444</td>
<td>0.27777</td>
</tr>
<tr>
<td>RL3</td>
<td>0.194444</td>
<td>0.27777</td>
</tr>
<tr>
<td>RR3</td>
<td>0.194444</td>
<td>0.27777</td>
</tr>
</tbody>
</table>

CREATE "PROJECTED" UNIT LOAD SETS

Snow loads are always projected to the tributary ground surface and include the roof surfaces for this model. The 6 unit load projected load sets are created all in one operation. Select nothing and use Set loads. Open the Projected tab. Set Y-Negative, Plates only, and Use compensation. Press the Unit sets button. Select the 6 roof regions from the list. Choose or create a folder to place the projected unit load sets (preferably an empty one). These 6 load sets correspond to a 1 psi condition on the 6 roof surfaces projected to the ground plane and oriented positive downward as the dialog is set up.

ASSEMBLE THE SNOW LOAD SETS

A snow load condition is created for each column of the above table. Open the File tab in global loading dialog. Use the Assemble button. Navigate to the unit load folder. Add 1 more column to the table so that there are 2 conditions. You can rename the columns (cases) to $S_b$ for balanced snow and $S_u$ for unbalanced snow corresponding to the 2 cases for which we have pressure distributions in the above table. Fill in the values.

<table>
<thead>
<tr>
<th>Load set</th>
<th>$S_b$</th>
<th>$S_u$</th>
</tr>
</thead>
<tbody>
<tr>
<td>UnitP_RL1.lsb</td>
<td>0.194444</td>
<td>0</td>
</tr>
<tr>
<td>UnitP_RL2.lsb</td>
<td>0.194444</td>
<td>0</td>
</tr>
<tr>
<td>UnitP_RL3.lsb</td>
<td>0.194444</td>
<td>0</td>
</tr>
<tr>
<td>UnitP_RR1.lsb</td>
<td>0.194444</td>
<td>0.27777</td>
</tr>
<tr>
<td>UnitP_RR2.lsb</td>
<td>0.194444</td>
<td>0.27777</td>
</tr>
<tr>
<td>UnitP_RR3.lsb</td>
<td>0.194444</td>
<td>0.27777</td>
</tr>
</tbody>
</table>

Right-click on the header and use the Save configuration item to save the layout in case you need to return and correct or repeat the operation.

You can save all the save these cases to a folder of their own. Exit and check the cases.

You can use Model/Resultant loads and generate a global total load table. (if you select the node in the floor center and opt to include moments then you can also see the overall moments as well as the resultant forces). The moment, $M_z$, is the overturning moment which is sometimes a requirement to determine for some structure types.

RESULTANT GLOBAL LOADS - about point (0.0, 0.0, 144.0) or Node 381

<table>
<thead>
<tr>
<th>Load set Fx</th>
<th>Fy</th>
<th>Fz</th>
<th>Mx</th>
<th>My</th>
<th>$M_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sb</td>
<td>0</td>
<td>-8064</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Su</td>
<td>0</td>
<td>-5760</td>
<td>0</td>
<td>0</td>
<td>-207354</td>
</tr>
</tbody>
</table>

DEAD WEIGHT

You can easily create the dead weight case. Use Set loads; open the dead weight tab choose Y negative for this model (typically the default); check the 'Use compensation' item for a slightly improved distribution. Use Apply dead weight. Open the File tab and save the case as D.lsb.
LOAD COMBINATIONS

These are some of the base loads. They would be further factored and combined with dead weight, snow loads, etc to create the design load combinations for analysis. Those combinations are also set up in the assembler by first putting all the needed loads in a folder together and setting up the factors. Here are only a few of the cases that could be generated for the ‘Allowable Stress Design’ (ASD) method.

You could have dozens or even hundreds of cases unless you apply experience and judgment to eliminate many combinations by inspection.

<table>
<thead>
<tr>
<th>Load set</th>
<th>Case 01</th>
<th>Case 02</th>
<th>Case 03</th>
<th>Case 04</th>
</tr>
</thead>
<tbody>
<tr>
<td>D.lsb</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.6</td>
</tr>
<tr>
<td>Sb.lsb</td>
<td>1</td>
<td>0.75</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Su.lsb</td>
<td>0</td>
<td>0</td>
<td>0.75</td>
<td>0</td>
</tr>
<tr>
<td>WXin.lsb</td>
<td>0</td>
<td>0</td>
<td>0.45</td>
<td>0</td>
</tr>
<tr>
<td>WXout.lsb</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0.6</td>
</tr>
<tr>
<td>WZin.lsb</td>
<td>0</td>
<td>0.45</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>WZout.lsb</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Use **Model/Resultant loads** and generate a table of global resultant loads.

Load set | Fx    | Fy     | Fz    |
---------|-------|--------|-------|
Case 01  | -12169| 0      | 0     |
Case 02  | -8778 | -1725  | 0     |
Case 03  | 3061  | -7273  | 0     |
Case 04  | 4081  | 1018   | 0     |

You can solve all the cases at once using **Solve/Static/Multiple**.

Once solved, you can display the maximum and minimum values for all grouped items for each case at once as well as across all cases using **File/Results/Review/Max-min**.

MAXIMUM/MINIMUM NORMAL STRESS, SN1 OR SN2

<table>
<thead>
<tr>
<th>Group name</th>
<th>Case 01</th>
<th>Case 02</th>
<th>Case 03</th>
<th>Case 04</th>
<th>Max/min</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base</td>
<td>7</td>
<td>201</td>
<td>127</td>
<td>267</td>
<td>267</td>
</tr>
<tr>
<td></td>
<td>-7</td>
<td>-201</td>
<td>-127</td>
<td>-267</td>
<td>-267</td>
</tr>
<tr>
<td>Corner posts</td>
<td>13</td>
<td>228</td>
<td>133</td>
<td>297</td>
<td>297</td>
</tr>
<tr>
<td></td>
<td>-30</td>
<td>-246</td>
<td>-134</td>
<td>-282</td>
<td>-282</td>
</tr>
<tr>
<td>Door base</td>
<td>6</td>
<td>1241</td>
<td>606</td>
<td>1494</td>
<td>1494</td>
</tr>
<tr>
<td></td>
<td>-11</td>
<td>-1234</td>
<td>-603</td>
<td>-1495</td>
<td>-1495</td>
</tr>
<tr>
<td>Door header</td>
<td>18</td>
<td>316</td>
<td>159</td>
<td>376</td>
<td>376</td>
</tr>
<tr>
<td></td>
<td>-2</td>
<td>-307</td>
<td>-144</td>
<td>-371</td>
<td>-371</td>
</tr>
<tr>
<td>Ridge beam</td>
<td>27</td>
<td>26</td>
<td>24</td>
<td>41</td>
<td>41</td>
</tr>
<tr>
<td></td>
<td>-54</td>
<td>-52</td>
<td>-36</td>
<td>-37</td>
<td>-54</td>
</tr>
<tr>
<td>Roof lateral</td>
<td>26</td>
<td>246</td>
<td>113</td>
<td>298</td>
<td>298</td>
</tr>
<tr>
<td></td>
<td>-22</td>
<td>-254</td>
<td>-111</td>
<td>-290</td>
<td>-290</td>
</tr>
<tr>
<td>Roof rafter</td>
<td>51</td>
<td>211</td>
<td>281</td>
<td>286</td>
<td>286</td>
</tr>
<tr>
<td></td>
<td>-107</td>
<td>-240</td>
<td>-297</td>
<td>-276</td>
<td>-276</td>
</tr>
<tr>
<td>Truss joist</td>
<td>136</td>
<td>193</td>
<td>236</td>
<td>212</td>
<td>236</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>-27</td>
<td>-233</td>
<td>-214</td>
<td>-233</td>
</tr>
<tr>
<td>Truss web</td>
<td>24</td>
<td>19</td>
<td>22</td>
<td>18</td>
<td>24</td>
</tr>
<tr>
<td></td>
<td>-27</td>
<td>-21</td>
<td>-27</td>
<td>-19</td>
<td>-27</td>
</tr>
<tr>
<td>Wall cap</td>
<td>36</td>
<td>622</td>
<td>304</td>
<td>720</td>
<td>720</td>
</tr>
<tr>
<td></td>
<td>-9</td>
<td>-604</td>
<td>-313</td>
<td>-744</td>
<td>-744</td>
</tr>
<tr>
<td>Wall stud</td>
<td>22</td>
<td>390</td>
<td>399</td>
<td>515</td>
<td>515</td>
</tr>
<tr>
<td></td>
<td>-63</td>
<td>-403</td>
<td>-414</td>
<td>-503</td>
<td>-503</td>
</tr>
<tr>
<td>Wall stud window</td>
<td>24</td>
<td>176</td>
<td>330</td>
<td>342</td>
<td>342</td>
</tr>
<tr>
<td></td>
<td>-80</td>
<td>-212</td>
<td>-359</td>
<td>-320</td>
<td>-359</td>
</tr>
<tr>
<td>Window header</td>
<td>28</td>
<td>31</td>
<td>32</td>
<td>35</td>
<td>35</td>
</tr>
</tbody>
</table>
You could also generate a similar table showing the stress ratios relative to the US Design code for wood construction.
The particular load condition applied to the model as presented in the sample is Case 04.

**GARAGE SPRING.FEM (GARAGE DOOR SPRING)**

This example makes use of the helix template and the component rotation utility.

**DESCRIPTION**

MATERIAL = STEEL
DIAMETER = 1.382 OD (1.25 to center of wire)
WIRE SIZE = 0.132
LENGTH = ~26.5 INCHES (of constant spacing turns) ~27 overall.
TURNS = 176.5 + 4 for the two double end loops
STRETCH = The spring is designed to stretch 42 inches.

**CONSTRUCTION**

Use the *Quick modeler* template for the helix

Center end node (-13.5, 0, 0)
Radius (R) = 0.625; Length (L) = 27.0
Turns (N) = 180.5; Segments; 4332 (24 per turn)
Choose axis normal to the ZY Plane

**PROPERTIES**

Set a property in the Library for a simple 'Line' element.
Set a property in the Library for the coil wire.
Select all elements in the coil an assign the wire property to them.

**FIX UP THE CENTERLINE**

Select one node on the coil (for reference) then select all the nodes on the centerline. One way to do this is to view the spring endwise in the Nodal Editor mode. Select some arbitrary node on the coil, then drag a small selection frame over the center nodes to select them all at once.

Right-click and use *Insert beams(s)* to connect them with beam elements. Go to the Element Editor mode and select all the new center elements and assign the Line property. (This feature ensures that the center reference nodes will rotate along with the end coils in the next operation.

**ROTATE THE END COILS**

Two full rings on each end will be rotated about nodes on the top and bottom

Left end: Select node 49 (on top of the coil near the left end) and select all element to the left of node 49 including the center line type elements

Choose *Utilities/Constructs/Rotate* and rotate the selected coils by -50.5 degrees on the global Z axis (rotation node 49)

Light end: Select node 4285 (on bottom of the coil near the right end) and select all element to the right of node 4285 including the center line type elements

Choose Utilities/Constructs/Rotate and rotate the selected coils by -50.5 degrees on the global Z axis (rotation node 4285)
SPECIFY RESTRAINTS INCLUDING SPECIFIED DISPLACEMENT

Restraints set for a specified displacement of the right end by 42 inches. (This is usually about 1/2 the Garage door height of 84 inches).

Left end (node 13): X fix, Y fix, Z fix, Xrot fix; Xrot Free, Yrot free, Zrot free
Right end (node 4320): Disp X = 42; Y, Z, Xrot Fixed, Yrot free, Zrot free

GLIRDER.FEM (Girder plate model)

This is a model of a girder used to illustrate some differences between a slender beam representation and the full girder model as illustrated here especially in the presence of concentrated loads. For much more discussion of girder construction see CADRE Analytic white paper #008. The white papers are available at https://www.cadreanalytic.com for evaluators as well as licensed users.

The girder supports a 50000 pound concentrated load at the center. The girder sits on supports separated by 348 inches.

The girder section is shown below, however the web thickness at the ends (12 inch span) over the supports is doubled to 1.0 inches.

The basic section properties for use in beam analysis would be:

A = 77
Iz = 51345.67
Iy = 576.6
C = 31

Stress in the upper flange calculated by simple beam analysis would be:
M = (50000/2)(348/2) = 4.35E6 in-lb
S = Mc/I = 4.35E6(31/51345.8) = 2626 psi (or 2.626 ksi)

This model as local systems set up for the flanges and web so that one can examine planar stresses aligned with the girder systems. The stress Sx is in the aligned with the girder for all components.

We set the depth of the girder as 60 inches in the idealized model rather than 62 inches. That is the distance from the centroid of the upper and lower flanges.

Solving the model and checking the stress, Sx, for the upper flange gives a maximum normal stress of 5848.7 psi. This is more than double the value from the simple beam analysis.

The concentrated loads deform the web locally under the flange resulting in an additional bending of the upper flange which augments the compression already there on the upper surface.

There is also a significant increase in local web stress at the concentrated load.

The same issue is present at the reaction points as well but here we have doubled the web thickness. A doubler under the concentrated load may be useful as well.

Another factor to keep in mind when building a girder model is that the load and reactions are not on the neutral axis as would be assumed by slender beam analysis. You must be careful not to unduly restrain the model since this fact will cause the supports to tend to move laterally. Notice that the distance between the support points after solution is 348.015 inches.

There are other issues that must be addressed as well such as local buckling, stability and crippling of the web. These are usually (and more easily) addressed by keeping the geometry within the limits specified for concentrated loads in the AISC manual for steel construction.

There is a CADRE Analytic "White paper" concerning "Girder models" which covers these and other issues in much more detail.

**GLOVEBOX.FEM (LABORATORY GLOVE BOX)**

Typical static design requirements for these items include internal negative pressure, floor distributed loading, floor concentrated load, and dead weight. Acceptance criteria include surface and floor displacement limits and material strength limits. In many cases, seismic loads dictate tie down fittings.
This model is currently loaded with only negative internal pressure - 10 inches of water (i.e. -0.360912 psi)

The internal pressure load is applied using the "Normal" load distribution feature.

Note that overall, Fx, Fy, and Fz are zero as would be expected after applying an internal pressure in a closed chamber.

As presented, the floor accesses, arm holes, and the ventilation accesses on the back are covered with 0.375 stainless steel plates.

**OPACITY**

The transparent front Lexan sheet has an opacity setting less than 100% for visual effect. The actual appearance depends a lot on your video display and graphics adapters, etc so a little trial and error is usually needed to find the right value for the effect you are looking for.

**SURFACE DISPLACEMENT**

The floor centerline nodes are given a group name "floor centerline". To make a quick plot of floor displacement simply use Select/Nodes/By Group then chooses the 'floor centerline' nodes. Then use Results/Displacement diagram. You would typically want to examine Y vs X for the floor centerline. The front and back faces have similar group names to use for selection and displacement presentation. For those surfaces, Z vs X would be the display choice.

**STRESS**

The relevant stress to rank and color display is principal stress S2. When displaying stress maps of the surfaces, the presentation might be better if all surfaces have 100% opacity.

**GREENHOUSE.FEM (GEODESIC GREENHOUSE)**

This model is built over a guide frame generated by the Geodesic modeler under Utilities.

Some geodesics are based on hubs with struts while others such as this one are based solely on the panels themselves.

- Polyhedron type = Icosahedron
- Frequency = 2
- Class = I
- Eccentricity = 1
- Radius = 192
- Zenith = Y
- Faces = 40
- Struts = 65
- Surface nodes = 26
- One center node

With this frequency setting, an icosahedron has only two different triangle shapes.

Once the basic guide dome frame is developed, the two different panel shapes are made along with their 2x6 frame, connectors, and sheathing in separate fem files. They are each made within a guide triangle that matches the two types of triangles. Then they are simply merged onto the main frame at the appropriate locations. The main frame
properties are then set to nothing (line properties) and made invisible so that they will be available for reference but otherwise out of the picture. These panels are made relatively simple as an illustrative example but they can be more discretized, use more connectors, and otherwise be made as detailed as needed in their individual fem files before merging them onto the geodesic guide frame.

Panel domes may or may not have additional hubs since the attachments near the vertices essentially form a strong hub area around the geodesic node location.

Downloads placed on this model arbitrary using the auto load feature. Typically one would examine snow, wind and seismic load conditions. The snow pressure would be modified according to the slope that each panel makes with the horizontal. One can select a single panel then use the eye view tool and see read the slope for that panel.

**GUSSET.FEM (BOLTED BRACKET)**

This example shows how to analyze simple bolted brackets with a rigid bracket assumption in the manner normally achieved by classical methods.

Static load analysis of Help "Getting started" - Exercise 18

Note: This is a text book problem from "Strength of Materials". The text book answers for maximum and minimum resultant force in the rivets using the classical method is 3770 and 2273 respectively.

The bolts were described as 7/8 inch diameter but it doesn't matter since it is really just a static load problem. Classical methods assume plates infinitely rigid and the bolts elastic.

To duplicate classical assumptions, the bolts are represented in the model by soft spring supports; sufficiently soft so to render the plate relatively rigid. The actual displacement of the bracket is not important since the main aim is the loading of the bolts based on the layout geometry (as in classical analysis). In fact, the actual displacement of the bracket is enormous due to the arbitrarily soft springs.

Turn on the load vectors and also check View/Nodes/Display load values.

Solve and then use Results/Reactions and check the Resultant force column in the table of reaction forces to see that the CADRE Pro result is the same as the book answers.

See the Help, "Getting started", exercise 18 for the step-by-step details and construction methodology.

See the additional sample static model gusset2.fem for a realistic finite element analysis of the same bracket.
GUSSET2.FEM (BOLTED BRACKET)

The simple static load analysis example gusset.fem of Exercise 18 showed how to conduct a rigid body (classical) analysis of a bolted bracket with CADRE Pro mainly to determine bolt reaction forces (gusset.fem).

This model gusset2.fem is the same model as gusset.fem of exercise 18 but set up in a more realistic fashion to investigate the stresses in the plate itself rather than the bolts.

The load transfer spokes representing the bolt bearing loads are set up arbitrarily to represent the fastener rigidity. There are two definitions in the library that can be used; one for hard fasteners relative to the plate and one for soft fasteners relative to the plate. The soft definition provides answers very close to the classical analysis while the hard ones are quite different. Both are correct answers. With hard fasteners the load is distributed primarily by plate distortion while with soft ones the plate is relatively rigid and loads are distributed more by fastener flexibility (as with classical analysis).

There is little reason to model brackets in a more complex fashion than exercise 18, but it can be done as demonstrated here.

GUYPOLE.FEM (FRAME TOWER SUPPORTED WITH GUY WIRES)

The model is a 3600 inch (300 feet) tall light frame tower with guy wire supports. The purpose of this example is to illustrate preloads and rigging loads in a structure.

The vertical legs are standard 2.0 OD 0.156 steel tubing.
The diagonal elements are 1 x 1 x 0.25 angles
The horizontal side members are 1.5 x 1.5 x 0.25 angles
The guy wires are 1/4 inch 6 x 37 wire rope

All properties are imported from the US Steel section property database.

LOADS

\[ W = \text{Wind at 0.04 psi lateral \( X \) direction applied to projected area of frame elements} \]

\[ D = \text{Dead weight of all frame elements} \]

Total Load condition = \( D + W \)

The frame members are set up in the Element Library with their effective cross section widths provided for the F/L item. So when we create projected wind loads using the global loading module the result will be the projected frontal area of the beams at unit (1.0 psi) pressure. So we first create that load then factor it to 0.04 psi as required our criterion.

With nothing selected, use **Set loads**

On the global loading dialog, projected tab, \( X \) positive, Replace, Use compensation, and **Apply projected load**. This applies 1 psi to the projected area of elements (giving no credit for shadowing). Then, on the same dialog, open the File tab and use the **Factor** button. Enter a factor of 0.04 psi and press **OK**. This is the wind load result is about 2028 pounds.

Still on the File tab, use the **Save** button to save this as the wind load set.

Still in the global loading dialog, open the Dead weight tab, \( Y \) negative, Replace, Use compensation, **Apply dead weight**. Open the File tab again and **Save** this as the dead weight load set.

While still on the File tab, use the merge button to combine the saved wind load set with the dead weight set.

Press **OK** to leave the global loading dialog with the case \( W + D \) applied

\[ W = 2028 \text{ lbs} \]

\[ D = 8467 \text{ lbs} \]

**PRELOAD**

To assign the setting preload (tension is specified as a negative value), select the 4 guy wires in the Element Editor mode and use the **Preload** button on the editor panel (or use **Set preload** from the pop-up menu).

All wires are given a -625 pound setting (a negative setting provides tension for preloads).

**DEFINITIONS**

Initial setting preload: This is the value (-625) that you would assign to the pre-tensioned elements on the model, assuming that every node in the model were grounded and fully fixed to earth so that it would not flex. When the model is run with the Setting Preloads applied (and no external loads except those that would be automatically imposed while rigging the actual structure) the model flexes and as the preloads are released into the structure. As a result the preloads will reduce to some extent.

Rigging load: The final tension in the preloaded elements (guy wires) after solution with only the external loads that would exist at the time of rigging (i.e. dead weight) is called the "rigging load". In this case, if we run the model with the 625 pound tension preload without the external wind loads but with the dead weight applied we would read about 600 pounds of tension after solution. So 600 is the rigging load we would apply in the field and indicate on specifications while 625 (entered as -625 preload) is the setting load to which we initialize in the analytical model.

**CHECKING RIGGING LOADS**

If you have created and saved the two load sets separately as described above, then use the Global loading module to load just the dead weight set. Now solve and note that the tension in the cables is about 600 pounds after solution even though they were set up with a preload of -625 pounds. So 600 would be the rigging load.

**TAUT WIRES**

When you run the model (Standard Solution) with all the external loads applied, the wires on the downwind side do not become slack so the -625 pound setting is a sufficient preload setting; implying that the rigging load of 600 pounds tension would be satisfactory to prevent the wires from being slack during under the wind condition.
SLACK WIRES
If you select all the guy wires and use Set preload and change the setting to -575 pounds, then solve, the downwind wires become slack indicating that a -575 pound setting isn’t quite sufficient to prevent slack wires under the wind condition.

PROCESS OF ESTABLISHING RIGGING LOADS
In a typical design process, one would, by trial and error, develop a set of Setting Preloads that enabled the cables to remain in tension under the external load conditions. Once the setting preload is established. Remove the external loads (except those that would be imposed at the time of rigging in the field) and solve with the preload settings. Check the tension values. Then record those as the 'rigging loads' to provide in specifications.

ANALYSIS
On solution the tower displaces in an exaggerated fashion. You can remove the exaggeration by using the check box provided.

After solution, check the tension in the upwind and downwind guy wires.
Upwind: Axial load = 1158 pound tension
Downwind: Axial load = 42 lb tension
So the downwind guys still have some small load (42 pound) remaining.
With the model solved, use Results/Beam stress maximums.

MAXIMUM/MINIMUM NORMAL STRESS, SN1 OR SN2 - MAXIMUM OF EITHER END

<table>
<thead>
<tr>
<th>Group name</th>
<th>Max/Min</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Diagonal - L/R side</td>
<td>5857</td>
<td>-8116</td>
<td></td>
</tr>
<tr>
<td>Diagonal - f/b side</td>
<td>1078</td>
<td>-3171</td>
<td></td>
</tr>
<tr>
<td>Diagonal - horizontal</td>
<td>4589</td>
<td>-3286</td>
<td></td>
</tr>
<tr>
<td>Guy wire</td>
<td>23595</td>
<td>860</td>
<td></td>
</tr>
<tr>
<td>Horizontal - side</td>
<td>4715</td>
<td>-3330</td>
<td></td>
</tr>
<tr>
<td>Vertical</td>
<td>9278</td>
<td>-14256</td>
<td></td>
</tr>
</tbody>
</table>

The advanced solution solver can be used to check stability but ONLY as long as no tension element like the guy wires become slack during the process. If they do, the results are unreliable. Solve using the advanced method with about 10 increments and notice that no instability is detected.
HINGEDPANEL.FEM (HINGED PANELS)

This model is constructed in Exercise 24 in Help "Getting Started"

This exercise is a steel plate that is fixed along the left edge, hinged along a line through the center, resting on a pivot support along a vertical line in the right portion, and loaded in the global Z direction along the right edge.

This model to be constructed is a steel rectangular plate that is fixed along the left edge, hinged along a line through the center at 10 inches from the left edge, resting on a pivot support along a vertical line in the right portion (14 inches from the left edge), and loaded in the global Z direction along the right edge.

The exercise is worked in units of:

Force units as pound; Distance units as inch

The plate is modeled in the XY plane. The dimensions are 20 x 20 x 0.1 inch thick and the load on the right edge is 100 pounds total distributed along the edge.

Since a copy of the model (hingedpanels.fem) is already in your static sample files folder (Statsamp) you should set up a different folder to save the practice components and models that you create in this exercise or use a different name. In any case, if you accidently overwrite any of the sample files you can always restore those using Options/Restore sample files from the main menu bar.

It is recommended that you print out this topic to make it easy to follow along as you perform the exercise.

Use the Quick modeler to make the model (Utilities/Quick modeler).

Choose the template: plate mesh 3
In dialog enter:

- Lower left corner: 0, 0, 0
- Upper right corner: 20, 20
- Mesh size: 20, 20
- Check Systems

Press **OK**

Select the pertinent nodes and use *Show selected only* to make them the only ones visible.

![Diagram of a grid with labels and options for fixed restraint, hinge, pivot, and load line.]

It is best to assign properties to the surface before installing the hinge pin system since the stiffness properties of the hinge are estimated from the plate elastic properties and thickness. If not, then all the stiffness properties of the hinge will be zero and you will have to estimate them yourself.

Go to the Element library under **Edit/Library** on the CADRE Pro menu bar or choose the Library Tool on the main tool bar.

**Add** a library entry and change its type to *Kirchhoff plate*. Give it a name such as *Panel (Cold rolled steel)*.

Set the thickness, **T** to 0.1.

Select this item on the list, and then from the dialog menu bar choose **Materials**. Choose the *MatSampIPF.txt* materials list since it is in the same units as our model. Pick the cold rolled steel item from the materials list and use it to set the elastic modulus, **E**, and the Poisson ratio, **V**.

- **E** = 29.5E6 (pound per square inch)
- **V** = 0.287
- **T** = 0.1 inches

Set the beam plate selector tool to plates. Select all the plate elements.

Right click and use **Set properties** button and assign the steel property to the selected elements (entire surface).

Turn off the node selection circles so that only elements will be selected with the frame.
Now select all the elements on the right side of the hinge line. Select at least those abutting or connected to the hinge line.

It doesn't matter if more plate elements are selected as long as they are on the same side, so it is just as easy in this case to select the whole half of the panel.

Turn on the node selection circles.

With elements still selected, select also the nodes along the hinge line. They must be selected in order along the hinge line. You can select them one-by-one or select with the frame and use Select/Nodes/Re-order selection or (Ctrl J) and choose to re-order them by increasing (or decreasing) Y.

You can also select an additional node off the hinge line such as the far left middle level node. It will be used as the reference node for the hinge beams. Or you can wait and set the reference node after the fact.

With the above selection state, use Utilities/Insert special structures/Surface hinge.

At the query choose: Open ended line

Press OK and the hinge beam system is installed.

A confirmation message is displayed prompting you to:

- Set proper a proper reference node,
- Evaluate the stiffness settings for the hinge beam,
• And run the bandwidth manager.

VERIFY OR SET ORIENTATION

If you select and check the orientation of any hinge beam element (right-click and use Show orientation) you will see that the reference node is at in the plane of the surface and not on the hinge line. If else, you would need to select and set a proper orientation node for the hinge line.

With orientation verified you can see that Iy is the bending resistance of the hinge out-of-plane and Iz is the resistance to in-plane bending.

The stiffness of the hinge should be evaluated in light of the applied load levels and load directions so it is necessary to load and restrain the model before final confirmation of the hinge.

Go to the Nodal Editor. Select the left edge nodes. Use the Bound button and then use the Fix button to set fully fixed properties for all degrees of freedom on the embedded edge.

Select the nodes along the pivot support location at the right of the hinge line. Use the Bound button and set only the Z direction to Fixed, all else Free.

The right edge is loaded with -100 pounds in the Z (out-of-plane) direction. There are 21 locations so select the two end nodes. Use the Load button and place -2.5 pounds for FZ. Then select the remaining 19 nodes on the right edge and assign -5 pounds to FZ.

With orientation verified you can see that Iy is the bending resistance of the hinge out of plane and Iz is the resistance to in-plane bending.

Before solving, use Tools/Bandwidth manager.
Solve using Static/Standard and notice how the hinge performs. There is no moment transmitted across the hinge line.

**EVALUATE HINGE STIFFNESS**

Typically this hinge structure is just an artifice used to remove the tendency for moment to be transferred across a line in a surface. In some cases, one may need to adjust these values based on the actual loading conditions. For example if there were significant forces tending to cause shear along the hinge line, then the axial stiffness (Area) may need to be increased. Also, if the surface displaced enough to open a gap under load, then the bending stiffness (Iy or Iz) may need to be increased in one or both directions.

Open the element library and review the new item Hinge beam that has been added.

The initial artificial stiffness of the hinge beam is set up based on a rod with diameter the size of the thickness of the selected plate. Then, it is adjusted to 10000 times stiffer in bending (Iy and Iz), and 1/10000 of the area (A). The torsional coefficient (J) is set to zero. The beam is fixed on the axis end (A-Ypin=1 and A-Zpin=1) and pinned on the origin end (O-Ypin=0 and O-Zpin=0). This is intended to be only an artifice that significantly resists displacement normal to the hinge line but allows free movement in torsion and only little resistance in axial displacement.

The size of the hinge pin (bending properties) must be carefully managed and perhaps adjusted so that it doesn't stiffen the plates in bending so much as to cause numerical issues, nor can it be so soft as to allow the plates to displace excessively relative to one another except in hinge rotation under the applied load condition.

The axial load must not be so stiff as to significantly resist the tendency of the plate to expand or contract along the hinge line if that is what the loading condition would tend to do. On the other hand, it should not be so soft as to allow significant differential motion of the left and right panels parallel to the hinge line.
Examining the model under load with extreme displacement exaggerations, one can see that these default values appear to be adequate for this model under this loading level and loading direction.

If the hinge were an actual hinge with real structural properties that could be estimated then you may need to adjust the properties to match the actual hinge which may also involve adjusting the free pin ends to some partial pin fixity.

**Some precautions**

The two panels have overlapping nodes at the hinge line. Those of one side are hidden to help prevent accidentally attaching them. One must take care not to un-hide the hinge nodes when using the Attach nodes feature or it will weld the panels together and the hinge will no longer function.

**Model checking anomalies**

If you use the Tools/Check model/Check for anomalies feature it will identify 42 'collocated nodes' and 20 potentially invalid property assignments. The 42 collocated nodes are intentional in the model and not an error. Overlapping nodes are detected even if hidden.

Also, the 20 invalid property assignments are just the fact that J was set to zero for the torsional stiffness. That too is intentional for this particular model. DO NOT USE the Repair button!

If you use Tools/Check model/Check elements properties you can choose to ignore the beam torsion stiffness in the invalid property count and then the invalid property count will be zero.

**HINGE LAYOUT**

If you wish to study how the beams are set up select the right half of the plate elements and use Utilities/Constructs/Translate. Do NOT check the detach option and enter a value of 2 for the X distance. You can see the X pattern of beams between the plates.

Select a couple of them and use Show direction.

The beams are actually parallel and the axis end of the beam (fixed end) is on the originally selected half of the panel. You could have selected the left half at the start and the performance would be identical but the direction shown above would be reversed.

Another sample model with a closed loop hinge is demonstrated by Drum1.fem.

**HOLES.FEM (MODELING PLATES WITH HOLES)**

This model illustrates ways of making holes in plates by different methods and with different features.

The basic plate is made with the Quick Modeler 'Plate mesh 3' option. The plate is 20 x 10 with 40 x 20 divisions.
The hole in the middle was made by first cutting a 5 x 5 rectangle out of the basic plate center. Then, separately, a hole model was made using Utilities/Hole model. The size specification for the model was 2.5 x 2.5 with a 1.75 radius. A 5 x 5 division with a radial division of 5 and with the log discretization option were the discretization choices. The hole model was saved, the original plate model with the open 5 x 5 area was reloaded and the hole model was merged in using the 'Merge' option.

Alternatively, you can directly insert the middle hole template into the model by first selecting the three corner nodes of the opening and using **Utilities/Hole model**.

![Diagram of a plate with a hole](image)

Making holes with the Hole utility will create a relatively good discretization around the hole boundary and you can check aspect ratio as you develop the model.

The specification for the middle hole was as follows:

**HOLE MESH UTILITY SETTINGS**
- Centerpoint = 0,0
- Upper right point = 2.5,2.5
- Hole radius = 1.75
- Vertical divisions = 5
- Horizontal divisions = 5
- Radial divisions = 3
- Discretization method = Mod linear
- Quadrants = 4
- No regional system
- Evenly distribute rim nodes

**RESULTING CONFIGURATION**
- 161 nodes
- 40 rim nodes
- 161 nodes
- 240 plate elements
- Aspect ratio (Maximum) = 1.777

This method can create relatively good discretization around the hole boundary and you can check aspect ratio as you develop it.

The hole on the left side of the model was made using the simple (somewhat crude) "Cut hole" option under Utilities/Constructs. First the center node is selected in the plate, then the 'Cut hole' command brings up a dialog. This removes elements within the radius then pushes and pulls nodes close to the rim into the rim. After creation, some of the closer nodes to the circle on the outside of the rim were strategically selected, dragged, and dropped into adjacent rim nodes. Finally, the **Attach nodes** and **Delete invalid elements** features were run to clean up the result.

The hole on the right was made just like the one on the left except the option to leave the hole covered was checked. The cover part in the hole on the right is automatically provided a group name, so you can use **Select/Elements/By Group** to easily select the cover elements and assign different properties. Here the properties are left identical to the plate so from a stress standpoint the right hole area is more or less treated as just a continuous part of the plate as if the hole did not exist.

The model is restrained to provide a minimum of restraint influence by using nodes on lines of symmetry and restraining only in directions that would not be expected to move anyway. You will see some reaction vectors at these points but if you check their values you will see that they really are very small (essentially zero) and simply due to internal numerical accuracy. If you don't want to see these reaction vectors, just go to **View/Nodes/Show Reaction Vectors** and uncheck that menu item.
For the sample solution the plate was loaded with 400 lb/in in tension applied to the nodes on each end.

As can be seen, the covered hole is essentially non-existent from a stress point of view.

**HOTSPOT.FEM (CENTRALLY HEATED DISK)**

The disk is 300 mm diameter by 3mm thick, with a 60 mm diameter heated center. The differential temperature is 120 degrees centigrade

Uns in Newton, millimeter (Megapascal)

**CONSTRUCTION:**

*Quick Modeler* template - "Spheric" with zero height

-- XY plane
-- Plates only
-- Center at (0, 0, 0)
-- R = 150, H = 0
-- Rings = 10, Sectors = 8, Boundary adjusted rings = 2

Material is AISI 301 stainless steel

\[
E = 186200 \\
G = 73080 \\
\text{Force density} = 7.763E-5 \text{ (Newton/mm}^3\text{)} \\
\text{Expansion coefficient} = 1.728E-5 \text{ per °C}
\]

Two Library entries are set up only for the purpose of setting a different color identifying regions.

The central cord is selected and given a group name "Hot zone" and the outer region is given a group name "Cold zone" for later use in evaluation after solution.

The disk is minimally restrained at the extreme top, bottom, left, and right, to simulate a free boundary evaluation.

The temperature differential +120 degrees and expansion coefficient are set for the center region by selecting the 'Hot zone' and using the Element Editor "Preload" button (or Set preload from the pop-up menu).

**CLASSICAL SOLUTION:**

Classical stress in central heated core area

\[
S = \alpha E \left( \frac{T}{\rho} \right) = 193 \text{ Mpa}
\]

This classical formula assumes a thin plate with a small core radius relative to the disk radius.

**CADRE PRO SOLUTION:**

Solve. Then...
Click on various plates and using the Eye view tool to review the stresses in the core area. You can see the Von Mises stress ranges from 194.7 to 199.5 Mpa. The more methodical method is to select the core area only by using Select/By group or Ctrl_G and choose Hot zone. Then use Results/Plate stress.

On the dialog, you can use the default 'centroid' location and 'front side' but since these stresses are in-plane it won't make any difference. Those settings affect the way the bending component of stress is included and there is no bending in this particular solution.

The initial display is with the selected center 'Hot zone' only. Scroll down to the bottom and read the Von Mises stress in the "Ave(weighted)" row. The average value of stress weighted by plate areas reads 198.06 Mpa.

REFINEMENT:

If you open the element editor mode and select and Divide the entire plate using the "Divide by 4 - sides divided by 2" which maintains similar triangles, and resolve (after using the bandwidth manager!), you will get an average value of 189.37 Mpa in the "Hot zone".

The CADRE Pro solution may be a better estimate of stress than the classical formula since the CADRE Pro solution does not have the geometric limitation of the formula which assumes the disk radius is large relative to the core radius.

HYDROSTATIC2D.FEM (HYDROSTATIC LOADING)

This sample demonstrates the use of the "Hydrostatic load distribution" option as applied to beam type elements. This is an elliptical underwater passage modeled in 2D with beam elements to represent a single strip around the perimeter of the surface.

Major diameter is 120 inches
Minor diameter is 90 inches

The passage is submerged so that its top is at 75 inches below the surface. Consequently, the center is 120 inches below the surface and its bottom is 165 inches below the surface.

The task is to determine the buoyancy of this unit section of the passage and the internal load in the passage wall. The density of this water is 0.03628 pounds per cubic inch.
CONSTRUCTION

The model is quickly constructed in the Quick modeler using the Ring template with radius of 60 and an eccentricity of 0.75.

ELEMENT PROPERTIES

The section properties are for a corrugated steel sheet unit width (1 in amp - 4 inches/cycle - 1/8 inch thick).

The properties were determined by modeling a single unit strip of the corrugated sheet in another application (CADRE Profiler).

Since the strip is 1 unit wide the pressure (psi) is the same numerical value as the force per unit length acting on the strip.
RESTRAINT

One node at the bottom of the tank is selected for restraint in all 6 degrees of freedom.

LOADING

Loads are applied using the Global 'Hydrostatic load' feature as it applies to beam elements. This applies loads normal to the beam element segment (and in its local XY plane as determined by its orientation node).

The load is accumulated around the perimeter according to the intensity parameter (pressure) multiplied by the relative elevation of the centroid of each individual element.

To load the model, select all the elements in the model (since all are below the surface).

Use Set loads from the pop-up menu.

Open the Hydrostatic tab.

Choose Beams only if not already selected.

Set the intensity as -0.03628 lb per cubic inch

Set the Direction of increasing load as Y, Negative.

Set the Zero load level to 120 (surface of the water relative to the global origin).

Choose loads in Both directions or Neg (-).

The positive sense for hydrostatic loads on beam elements is always away from the element’s orientation node.

We could have used Y positive for the direction of increasing load, and then used a positive intensity (+0.03628) and the result would be the same. The applied forces at every node should be inward and increasing in magnitude.

Use the Apply hydrostatic load button.

The resulting Net force on the model is Fy = 307.07 pounds (upward)

Use OK to apply the load.
**SOLVING**

The structure is restrained for stability at a single point on the bottom. After solution, the reaction at that point is 307.07 pounds holding the structure down. In other words, the model has a buoyancy of 307.07 pounds per unit length of passage (assuming that the pressure is exerted only on the outside and that it is weightless).

One could easily generate the dead weight set and merge it as well to determine net static loads.

**CHECKING**

The buoyancy of a unit length calculated from simple physics is the weight of the unit width of passage as if it were water (i.e., weight of the water displaced). The volume of a unit width is \( \pi a b \times 1.0 = 3.1416 \times 45 \times 60 = 8482 \) cubic inches. At a density of 0.036268 pounds/cubic inch the buoyancy would be 307.63 pounds which compares well to the 307.07 reaction force and uplift force calculated from the model.

You could get as accurate as you wanted simply by increasing the number of nodes used to define the model.

Use **Results/Axial load** to check of the internal loads. Scroll to the bottom of the list. This shows that the corrugated wall material has a load (in compression) on the order of 160 (minimum) to 200 (maximum) pounds. That is, pounds per inch since this is a 1 inch wide strip of the passage.

**PARTIALLY SUBMERGED**

Check the case where the passage is located so that 30 inches of it is above the surface.

In this case the point of zero load is at 45 – 30 or 15 inches above the global reference. All else remains the same, however be sure NOT to use the *Both* option for the application control. Use *Neg (-)* so that loads are only acting when below the surface.
With this selection loads are not applied when they would act outward, only when acting inward (negative).

The buoyancy in this case is 217.6 pounds.

For other examples of hydrostatic loads on a 3D plate type model see the sample files Watertank.fem, vat.fem, and tankinertia.fem.

**HYPERBOLICPARABOLOID.FEM (HYPERBOLIC PARABOLOID ROOF STRUCTURE)**

This example is to illustrate the surface shaping utility *(Utilities/Constructs/Shape surface)* that can deform a flat surface into several types of parabolic related shapes based on the mathematical equation.

\[ W = W + AU^M + BV^H \]

An example of a real structure in this general form is the Warsaw Ochota railway station. The unique nature of the hyperbolic paraboloid structure is that all the roof beams from edge to edge in both orthogonal directions are absolutely straight. Even though the roof is a double curvature, there is not a single curved beam in the roof.

This sample structure is 1200 inches on a side as projected onto the ground.

The sample model structure is square and 1200 inches on a side as projected onto the ground. The mean roof height is 300 inches. The high corners are at 600 inches above the ground and the low corners are at ground level. The coordinate system is set up with the origin at the center of the roof but on the ground. The model is first built up as a flat square roof, then re-oriented 45 degrees to make a diamond shape, then deformed according to shape of a hyperbolic paraboloid.

**CONSTRUCTION DETAILS**

Start with the *Quick Modeler* utility

- Choose the Wire frame template
- Choose: XZ plane
- Set Corner node 1: -600, 300, -600
- Set Corner node 2: 600, ------, 600
As initially generated, the wire frame grid will be in the XZ plane with corner nodes (X,Z) = (-600, -600) and (+600, +600) and at elevation Y = 300. Although it appears initially as an edge-on view in the XY screen viewpoint setting, it would look like this if viewed from above.

REORIENT

While in the default XY screen view, the model initially appears with an edge-on view on the screen (Y upward, Z out of the screen). Leave it that way (in the XY screen viewpoint) for now and use the Rot Y scroll tool to rotate the model carefully to exactly +45 degrees (hint: check the angle on the status bar).

With the model exactly at that orientation (still an edge on view and rotated Y=+45 degrees), use Utilities/Constructs/Reorient to re-orient the model geometry permanently to this position.

Now rotate the model to look down on the XZ plane (i.e. depress Ctrl key and use the tool on the tool bar)

Check the corner nodes and notice that they are now at 848.5281 inches from the center with our new global coordinate system for this model and the roof shape is now a 'diamond shape' rather than a square, relative to the new global system.

ADD COLUMNS

Next select the edge nodes around the surface. They automatically have a group name (EDGE) provided by the quick modeler template so use Select/Nodes/By Group and choose the EDGE name to select them all at once.

When selected, use Utilities/Constructs/Copy nodes to in order to extend beams from a set of selected nodes.
Check the option to include connecting beams and enter -300 for the relative location for the nodes (i.e. on the ground). Choose OK and the new columns are installed and extend from the roof to the ground. Rotate the screen view to see them better.

DEFORM THE ROOF

In this orientation, the general equation for any type of paraboloid (with an appropriate origin at the center of the surface) is:

\[ Y = ax^2 + bz^2 \]

With \( a \) and \( b \) of opposite signs, this produces a hyperbolic paraboloid shape.

The building front is to the on the positive X axis. Given that we want the building front and back corners (left and right corners respectively in the picture) deformed upward to 600 inches (300 inches above the mean height, you can calculate \( a \) and \( b \) as follows:

At \( Z = 0 \) and \( X = 848.5281 \) we know that \( Y \) must rise 300 inches above the roof mean height (\( Y = +300 \))
so; \( a = 0.000416667 \)

At \( X=0 \) and \( Z = 848.5281 \) we know that \( Y \) must drop 300 inches to ground level (\( Y = -300 \))
so; \( b = -0.000416667 \)

Select all the nodes in the roof (and only in the roof).

Then, use Utilities/constructs/Shape surface.

Choose the parabolic Cartesian equation: \( W = W + ax^2 + bxz \)

In the dialog set the dependent variable \( W \) as the \( Y \) coordinate. Set \( U \) to \( X \) and set \( V \) to \( Z \); so that these match the equation shown above. Ensure that the exponents \( M \) and \( N \) are both set to 2.

Enter the calculated parameters \( A = 0.000416667 \) and \( B = -0.000416667 \).

Press OK to re-shape the roof to the desired shape.

The corners that bent down to the ground left zero-length columns and some overlapping nodes at those points. Clean up by using:

Select all nodes and use Results/Plate stress maximums to re-move the overlaying nodes at those corners.

Use Tools/Delete duplicate beams (which deletes any invalid zero-length elements as well).
CLAD THE ROOF

Go to **Utilities/Clad** can be used to clad a wire frame grid with surface plate elements. Enter point coordinates far below the model (say 0, -1000, 0) for the orientation. Choose the option 'include quadrilaterals'. Leave everything else at default. Use **OK** to place plate elements on the roof.

FINISH CONSTRUCTING

The remainder of the construction is the same as for all models.

1. Set the final grid and extents using **Edit/Grid and extents**, using ‘From model’
2. Set proper orientation nodes for all beams
3. Set properties and attributes in the element library and assign
4. Use **Tools/Check model** for final verification
5. Use **Tools/Bandwidth manager** for final efficiency

ALTERNATE STARTING MODEL

If the dimensions from side-to-side were different from front-to-back then one would build the diamond shape directly:

1. Use **File/New model** with dimensions capable of enclosing the diamond shape.
2. While in the nodal editor mode, set 4 node points at the 4 corners; all at the roof elevation.
3. Select these four points (counterclockwise or clockwise around).
4. Use **Insert mesh** with X and Y divisions set as desired for beam spacing.
5. This provides the flat diamond mesh and all else proceeds as described above.

ALTERNATE CONSTRUCTION METHOD

To build a simple hyperbolic paraboloid shape for the roof, one can first set 4 corner points at their final positions; two high and two low. Then select them counterclockwise around and use **Insert Mesh** with the **Wire frame quad** option. This method avoids the math and develops the curved roof shape shown above.

**IMPACTBEAM.FEM (IMPACT EXERCISE)**

This is a wooden beam, 120 inches long by 15 inches deep and 5 inches wide. A 48 pound concentrated weight is dropped from a height of 12 inches above the static beam (considered weightless). (The beam's material properties are \( E = 1.5 \times 10^6 \), \( G = 0.5 \times 10^6 \), Density = 0). This model is developed in Exercise 20 in Help, "Getting started".

Beam properties generated from the basic shapes generator are:

Rectangular bar 15 x 5

<table>
<thead>
<tr>
<th>Stiffness properties and loading parameters</th>
<th>E</th>
<th>G</th>
<th>W/L</th>
<th>F/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>1500000</td>
<td>500000</td>
<td>1.73625</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>A</td>
<td>Ly</td>
<td>Iz</td>
<td>1406.25</td>
<td>493.7431</td>
</tr>
<tr>
<td>75</td>
<td>156.25</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The total energy absorbed by the structure by the time the weight comes to rest at the maximum beam deflection state is:

\[ E = W(h + d) \]

Where \( h = 12 \) inches but the displacement \( d \) on impact is unknown.

**ITERATIVE SOLUTION**

1) Estimate \( d \) as 0.1 inches
2) Energy = 48(12 + 0.1) = 580.8
3) Apply \( d \) to the center node as a specified displacement.
4) Solve and read total energy under \textit{Model/Gross properties} = 293.0

Continue iterating

1) Estimate \(d\) as 0.13 inches
2) Calculate energy = \(48(12.13)\) inches = 582.2
3) Apply \(d=0.13\) to the center node as a specified displacement.
4) Solve and read total energy under \textit{Model/Gross properties} = 495.1

1) Estimate \(d\) as 0.14 inches
2) Energy = \(48(12.14)\) inches = 582.72
3) Apply \(d\) to the center node as a specified displacement.
4) Solve and read total energy under \textit{Model/Gross properties} = 574.2

1) Estimate \(d\) as 0.1402 inches
2) Energy = \(48(12.1402)\) inches = 582.73
3) Apply \(d\) to the center node as a specified displacement.
4) Solve and read total energy under \textit{Model/Gross properties} = 575.2

1) Estimate \(d\) as 0.1410 inches
2) Energy = \(48(12.1410)\) inches = 582.77
3) Apply \(d\) to the center node as a specified displacement.
4) Solve and read total energy under \textit{Model/Gross properties} = 582.5 (close enough)

So, \(d = 0.1410\) is the maximum impact displacement.
Moment at this state: \(M = 24790\) inch pounds
Stress = \( \frac{M}{Sz} = \frac{24790}{187.5} = 132 \text{ psi} \)

CHECK BY CLASSICAL ANALYSIS

DISPLACEMENT UNDER STATIC LOAD OF 48 POUNDS:

\[
d_s = \frac{WL^3}{48EI} = 0.0008192
\]

CLASSIC DYNAMIC FACTOR EQUATION:

\[
d = \left( \frac{96EIh}{WL^3} \right)^{1/2} = 171.16
\]

\[
d = 0.1402
\]

JUSTIFICATION OF DIFFERENCES

The classical analysis assumes the beam displacement \( d \) is small relative to the height \( h \) and this additional potential energy is ignored. This is realistically accounted for in the FEA analysis and so it is slightly more accurate.

**INCLINEDREACTION1.FEM (Slide bar type)**

CURVED BEAM ON AN INCLINED REACTION PLANE

This illustrates one method of creating sliding restraints on inclined planes.

**IDEALIZED MODEL**

The ring is 48 inch radius and spans 150 degrees to a 30 degree inclined plane where the end is free to slide along the plane but must remain on the plane.

**PROPERTIES**

Ring is 0.5 inch round aluminum bar.

The sliding restraint is a fictitious bar made very stiff in bending (relative to the flexible ring) and very soft in axial stiffness (relative to its bending stiffness). Here it is defined as steel and given \( I_y \) and \( I_z \) as 10 and the area is given as 1E-10 (i.e. almost zero but still non-zero).

One must be careful of numerical instability with these wide differences in stiffness. Watch the condition number!

Nodal displacements at the plane contact point

\[
X = 2.6979E+00 \\
Y = -1.5577E+00
\]
See model "inclinedreaction2.fem" for an alternate method.

**INCLINEDREACTION2.FEM (Pivot bar type)**

**CURVED BEAM ON AN INCLINED REACTION PLANE**

This illustrates one method of creating sliding restraints on inclined planes.

**IDEALIZED MODEL**

The ring is 48 inch radius and spans 150 degrees to a 30 degree inclined plane where the end is free to slide along the plane but must remain on the plane. The ring has a 5 pound point load applied parallel to the inclined plane.

**PROPERTIES**

1) Ring is 0.5 inch round aluminum bar.
2) The pivot restraint is a fictitious bar made very stiff, both axially and in bending (relative to the flexible ring). Here it is defined as steel and given Area, Iy, and Iz as 10.

One must be careful of numerical instability with these wide differences in stiffness. Watch the condition number!

Nodal displacements at the plane contact point:

- X = 2.6998E+00
- Y = -1.5587E+00

See model "inclinedreaction1.fem" for an alternate method.
**JUG.FEM (Modeling demonstration)**

This is not so much a practical structural model but provided for the purpose of demonstrating the curve fit, clone, clad, merge, and other modeling utilities as well as to demonstrate how to handle plate orientation for convoluted surfaces. Using these utilities the model is constructed, loaded, and solved starting with just 5 measured points on the surface.

The model is a jug type container made of 0.125 inch copper plate. Measurements of diameter and height are taken at 5 points vertically on one edge of the surface profile. The bottom flat surface is 3 inch in diameter.

Assume the problem is to determine the stress in the container when corked with 10 psi internal pressure.

Start with a **New** static model

Lower left: -3, 0, -3
Upper right: 3, 8, 3
G=0.5

Set these 5 points taken from measurements:

1: -1.5, 0, 0
2: -3, 2, 0
3: -0.5, 7.5
4: -1.0, 8.0
5: -1.5, 7.5

Select points 1 thru 5 in order.

Black and white mode (i.e. **View/Black on white**) was used for many of the following images where it may improve clarity when presenting on white paper.

Use **Utilities/Insert Special structures/Curve fit**.
Typically a couple of trials may be needed to achieve a reasonable point density and curvature. Here we settled on about 5 segments per interval with curvature parameter of 0.5.

With 5 points per interval and 4 intervals to fill, this would be a total of 20 segments distributed in proportion to relative size of each interval but always rounded up to a minimum of 2 per interval.

This results in a total number of 21 beam segments in this model.

Set one additional node (#23) at point (0, 0, 0)

Turn off the node circles and drag a frame to select all of the profile lines.

Turn on the node circles and drag a frame to select all the profile nodes.

With both nodes and elements selected, use Utilities/Clone. For a good quadrilateral aspect ratio about 36 sectors around the circumference would work fairly well. That would be 10 degrees per sector around a Y directed axis centered on node #23.
All nodes should be attached and a lateral beam should interconnect each copy at all selected nodes.

The final wire mesh should look like this.

There will be 21 duplicate elements along the vertical closing seam.

Use **Tools/Delete duplicate elements** to remove them.

Use **Utilities/Clad**.

Each quad is filled with 2 triangular plate elements.

The orientation is only partially finished as there is no single point that could orient the plates for this convoluted surface.

The point 0, 2, 0 works for the surface below the neck but the upper neck and lip sections will need special attention to orientation to be consistent inside body surface of the model.
The clad model should look like this (in color view mode) with the wire frame mesh still applied in addition to the cladding.

Remove the wire frame mesh.

Go to the Element editor mode E. Set the plate beam selector to select beams .
Select all the beams and Delete them using the pop-up menu.

To complete the orientation you can carefully select parts of the model and hide others. It is especially useful to give each portion a different group name so you can access it again more easily if needed.

To start, you can select all below the neck and lip and give it a name such as Body then hide that part.
Next you can use **Select/By radius** to select only the outer flared portion and give it a name *Lip*.

Hide it and then select the remainder and give it a name *Neck*.

The Body is already oriented properly by the cladding operation but the neck and lip need to have the inside and upward set as the *back side* to be consistent with the internal body surface.

Select the lip section only and orient it to location (0, 10000, 0).

Select the neck section only and orient it to location (0, 7.5, 0).

**INSTALLING A FLAT BOTTOM**

The profile line was not extended to the center line in order to avoid triangular plates with excessive aspect ratios. Instead the bottom circular disk area was left open so that it could be filled with a Quick Modeler 'Spheric' with good aspect ratio.

Go to the Quick modeler. Create a *Spheric* model with 36 supports to match the number of meridians on the model.

The above settings create reasonable aspect ratio triangles yet still match the support requirement.
Use the **Merge** method to attach this *Spheric* template model to the bottom of the main structural model.

Select and show only the bottom tier of clad plates on the jug, then orient to the XZ view. Select 3 nodes on the rim of the open space; one upward to orient; then two at opposite sides across the full diameter.

Choose Merge from the pop-up menu. Choose the *Spheric* model and set similar points on it.

Select **Continue** to install the flat disk into the jug model.

Finally, you can select the bottom surface and orient it to point (0, 2, 0) to designate the inside of the bottom as the back side consistent with the remainder of the surface.

**SOLUTION:**

The purpose of this sample is to demonstrate some of the modeling features but the model can be completed with material properties, loaded, and solved as below.
The library is set up with a copper material at 0.125 inches thick.

The section from the neck center and below (assumed corked) is subjected to a 10 psi pressure for the loading condition for all plate elements below the throat (7.5 inches). You can use the load arrow view to verify that pressure is outward and this verifies the proper front/back orientation of the plates.

The maximum von Mises stress is 1185 psi on the bottom surface.

*LAMINATEDPLATES.FEM (Laminated wafer)*

Silicon 0.02 mm thick silicon plate laminated to 0.1 inch thick phenolic plate.
DIMENSIONS

TOP VIEW

```
20 mm
10 mm
```

THICKNESS VIEW

```
0.2mm
1.0mm
0.6 mm
```

IDEALIZED MODEL

- Silicon layer 0.2 mm
- Phenolic layer 1.0 mm
- Rigid rods 0.6 mm

MATERIAL PROPERTIES:

One must always consider the limitations of the software, especially for these types of materials.

CADRE Pro is linear elastic and assumes isotropic materials. The materials in this model are anisotropic and are not linear over a large range of stress.

- Assume we have an effective secant modulus relative to the expected stress range in the direction of importance.
- Assume that biaxial differences are is not too large and also choose a modulus related to the direction of importance.
- Assume these are the properties at the temperature of interest.

Silicon sheet

\[ E = 129600 \text{ Mpa} \]
\[ V = 0.27 \]

Phenolic sheet

\[ E = 20680 \text{ Mpa} \]
\[ V = 0.30 \]

Just for rough sizing, we set the total area of connector beam cross sections to be 200 mm\(^2\) which is the same as wafer area. So the connector beams (0.6812 by 0.6812 square bar 0.6 mm long) are placed at every node and made of the phenolic. This is essentially rigid but not so much so as to cause numerical issues.

CONSTRUCTION

Use the **Quick modeler**

Plate 3 template:

Set XY plane.

Corners: 0, 0, 0 and 20, 10, 0

Divisions 20, 10

Check to set up a system

Outside of editing modes, without selection circles showing, select all elements with the frame selector. Turn on the selection circles and then select all nodes with the frame selector so that all elements and nodes are selected.

Use **Utilities/Clone**
Set Translate: X=0; Y=0; Z=-0.6

Choose to apply interconnecting beams at preselected nodes

Leave any rotations as ZERO

This creates a double plate model separated by Z=0.6 and with short beams between the plates at every node. These short beams should be rigid.

**ELEMENT LIBRARY**

1) Set up a Kirchhoff plate element with the above silicon properties, set T=0.2 mm; leave W/A and F/A zero
2) Set up a Kirchhoff plate element with the above phenolic properties, set T=1.0 mm; leave W/A and F/A zero
3) Set up a standard beam element (separator bar) with silicon properties; square bar at 0.6812 x 0.6812 dimensions.

Note: the spacer beam is essentially rigid but not so rigid as to result in numeric issues. One way is to make it from the more rigid material with an area that, when summed over all 431 spacer beams, roughly equals the size of the panel. Still, check the condition number after solution to determine the integrity of the solution.

For both the plate elements, set the plate opacity to about 75% and change the plate edge style to "Same as plate". This style shows a finite thickness so is still visible when viewed edgewise which will be essential in the next step.

**ASSIGN PROPERTIES**

You can select all beams and assign the spacer beam property.

You can place the model in the XZ plane. The surfaces are edgewise with the silicon (to be) on top.

In the Element Editor mode, drag a frame over the upper surface and assign the silicon property to it.

In the Element Editor mode, drag a frame over the lower surface and assign the phenolic property to it.

**ASSIGN GROUP NAMES**

You can repeat the above actions on the surfaces and assign group names (perhaps "Silicon" to the upper surface and "Phenolic" to the lower surface. You can assign "Spacer" to the separator beams.

In the nodal editor mode you can drag a frame over the nodes in the upper surface and give them a name such as "SI". And for the lower surface use the name "PH". This helps when you want to hide and show various parts of the model.

**ASSIGN SYSTEMS**

If you created the model with the "Systems" item checked in the quick modeler than a regional surface system is already set up for the upper surface (System 1) and already assigned. Otherwise, you can create one using 3 corner nodes from the upper surface and using *Define system*.

**LOAD:**

10 Newton concentrated force at the center on the polycarbonate side toward the silicon side.

**RESTRAINT**

Restrain at the corners and at the load point from the phenolic side only.

Load point restrained in X, Y

Corners restrained in Z and just one of them also restrained in Y.

**SOLUTION:**

Before solving go to *Option/Settings* and open the Colorize tab. Check the *Transparent* option for the *Excluded element color*. With that option, you can stress color just one surface and examine it from any side since all other items will be hidden while stress coloring is in effect.

Solve the model. Then use *Results/Rank and color*. 
Rank only the silicon sheet item. It is better to rank on principal stresses or planar stresses rather than Von-Mises for brittle materials. We ranked on Planar stress, Sx which is oriented with respect to "System 1" and parallel to the long side of the rectangle.

The result is Sx = 46 Mpa in the silicon and the condition number is very good (~14). The stress in the phenolic is very low at about 5 Mpa.

We could increase the modulus of the rigid spacer by increasing orders of magnitude of their elastic modulus and notice that the condition number reduces noticeably and the stress level changes very little so the spacer rigidity is sufficient.

Notice that you can select and view the silicon sheet from either side while colored due to the 'Transparent' setting for the "Excluded element color" on the Colorize tab. It is still visible and grayed, check the Transparent setting and also make sure the hide/show button is off.

**LARGESTORAGETANK.FEM (Conical roof storage tank)**

**GEOMETRY**

- 56 ft high x 120 ft radius
- Conical roof 9.0 ft rise
- Wind girder 6 ft x 0.4375
- Wind girder flange 0.375 x 6
- Tank rim flange is 0.3125 x 8

This is a typical model of a conical (relatively flat roof) storage tank. It is loaded with a case of dead weight, wind, and snow on the roof and girder in addition to hydrostatic pressure from a full tank (specific gravity 0.8).
CONSTRUCTION

The tank wall portion was made from the Quick modeler "Cylinder" template.

- Plates only
- XZ plane
- Center 0, 0, 0
- Radius = 1440 (120 ft)
- Length = 672 (56 ft)
- Eccentricity = 1.0
- Sections = 7
- Sectors = 60
- Iso. triangles = unchecked

The girder was created by adding two radial nodes 6 feet outboard of the wall at one sector (Copy nodes relative with the radial option), inserting the single girder plate quad, support plate, and flange beam segment and then replicating those around the tank with the Clone utility.

The roof was made initially using a flat Spheric template.

- XY plane
- Plates and beams
- Center 0, 0, 0
- Radius 1440
- Height 0
- Rings 12
- Sectors 5
- No. Iso Adjust = 0

The roof was then modified extensively to add intermediate radials, delete certain intermediate beam elements to leave the purlin arrangement and finally the lateral beams were adjusted to be straight between posts to match the design drawings for the tank.

The flat roof was made conical using Constructs/Shape surface with a polar power equation to the first power.

Once one is familiar with all the templates and construction utilities these types of models can be created in a very short time with less tedium than with a CAD drawing. Since one builds it initially as an FEA model, setting properties and orientations as you go, it is essentially ready to solve when finally constructed.

FURTHER DISCRETIZATION

The model, as it appears here, is perhaps a little crude for final analysis but adequate for sizing structure. The model can be further discretized by dividing all beams by two, then all plate elements by 4 (each side of the triangle by two). Then use the feature Attach nodes to connect the new overlapping beam and plate nodes.

For round or curved plate surfaces there is one important caution that must be addressed with this discretization. The cylinder will have subdivided flat facets on the surface after the division which will seriously affect hydrostatic loadings. To correct this after the above discretization, make the tank wall the only visible component. Then select the tank wall and the center node and use Constructs/Force to radius to force all nodes in the tank surface to conform to the tank radius. This will ensure that the tank wall performs as a true shell under load.

LOADS

The model, as presented here, is loaded with dead weight, factored snow, factored wind, and full hydrostatic. The hydrostatic load overshadows the wind for this particular load case.
LEAFSPRING.FEM (LEAF SPRING MODEL)

The spring has 6 leafs each 2.5 inch wide and 0.375 inch thick. They are stacked together with spacers so that each is 0.4 inch center to center on the thickness. The spring span is 36 inches and the radius of curvature for the top leaf is 96 inches.

The angle formed by the top leaf about the center of curvature is

Arcsine (36/96) = 22.0243 degrees

The angle between each set of leaf connectors (straps) is:

22.0243/6 = 3.67072 degrees

CONSTRUCTION

Set up a new model with extents and grid at 0, 0, 0: 2, 2, 0: grid = 0.4

In the Nodal editor, turn on the snap tool and enter these 6 nodes for the fixed restraint end of the spring set:

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0.4</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0.8</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>1.2</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>1.6</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>2.0</td>
<td>0</td>
</tr>
</tbody>
</table>

Add elements between each of the 6 nodes. That is, 5 beam elements

Element Editor, draw beam elements between each of the first 6 nodes.

Right-click and use **Apply group name**. Give them a group name **straps**.

Now, to add a center rotation point for the upcoming replication use **Edit/Extents and grid** and change the setting to 0, 0, 0: 36, 98, 0: grid = 2. Set another node (node 7) at 0, 98, 0 (i.e. 96 inches above the last node)

Now, replicate and create the leaves:

Exit the Editor.

For the next operation you need to select all the elements and all of their nodes simultaneously.

Turn off the node circles and select all the elements.

Then (keeping the elements selected), turn on the node circles and select all the nodes in the strap area too (but NOT the center of curvature at node 7).
Use **Utilities/Clone** and set translation coordinates all to zero (i.e. 0, 0, 0).

Set the **Rotation angle** to 3.67072
Set the **Rotation axis** = Z
Set the **Rotation node** as 7 (i.e. center of curvature)
Set the **No. of Copies** = 6
Node numbering is **Sequential**
Check **Attach overlapping nodes**
Check **Add interconnecting beams**

Press **OK**

Go to the Element Editor mode. Select the appropriate leaf and connector elements that are to be removed to make the leaf arrangement and delete them.

Use **Select/Elements/By group** and select the **Straps** group.
Select the far right node of the upper leaf. Use the **Orient** button to set the **Strap** orientations to that node.

Use **Select/Elements/By group** again and this time select the **Unnamed** group.
Use **Apply group name** to assign a group name **Leaf** to these elements.

Use **Select/Elements/By group** again and select **Leaf**, (or just use **Reselect last**)!
Use the **Orient** button assign node 7 (center of curvature) to all the **Leaf** elements.

Exit the Element Editor mode, select all of the nodes, and use **Tools/Delete unused nodes**.

**PROPERTIES**

Go to the element library. Use the Basic Shapes item to make a leaf section property
Material Steel: $E=2.9E7$, $G=1.12E7$, Density 0.283
Use Structural Steel AISC from the MatSampIPF.txt materials file or just type in the material data.
Size: Height ($d$) = 0.375, Width ($b$) = 2.5,

The straps hold the spring leaves together but must allow them to slip relative to one another. This can be simulated (within small displacement theory) by simply pinning them about an axis normal to the spring assembly.

The straps are set up as steel rectangular bar section 0.1875 thick by 1.0 wide.

Material Steel: $E=2.9E7$, $G=1.12E7$, Density 0.283
Size: Height ($d$) = 1.0, Width ($b$) = 0.1875
Set the type of this section to 'Pinned beam' and change the pin settings to \( \{1, 0, 1, 0\} \) so that they will be allowed to slip relative to each other.

The effect of the pinned straps on the leaves can be illustrated schematically as follows where the pinned strap links allow the leaves to move horizontally relative to each other as they flex in the vertical direction.

In reality the leaves are in direct contact or sometimes separated by thin resilient spacers.

Go to the Element Editor mode.

**Select/Elements/By Group.** Choose the Leaf group. Use the Properties button and select the 0.375 x 2.5 property and assign it to all the selected elements.

**Select/Elements/By Group.** Choose the Straps group. Use the Properties button and select the 1 x 0.1875 property and assign it to all the selected elements.

Check the pin settings by selecting one or all of them. Right-click and use *Show pins*. They should show axis and origin Zpin axes oriented normal to the assembly.

**LOAD AND RESTRAINT**

Go to the Nodal Editor.

Select the nodes 1 through 6 on the left end and set the restraints to fully fixed in all 6 degrees of freedom.

Select the far end node of the top leaf and apply a load of -500 pounds.

Leave the Nodal editor mode.

**CADRE PRO SOLUTION:**

Solve using *Static Standard* and note that the loaded end moves out by 1.5 inches and down by -5.820.

Nodal displacements>
\[
\begin{align*}
X &= 1.5015E+00 \\
Y &= -5.8204E+00
\end{align*}
\]

Use the exaggerate check box to correctly display true displacement.

Solve using *Static Advanced* (use 1 increment and leave all items checked). Note that the end now moved by 1.6 inches outward and 6.206 inches downward which is probably a better solution.

Nodal displacements>
\[
\begin{align*}
X &= 1.6060E+00 \\
Y &= -6.2066E+00
\end{align*}
\]
Immediately after the advanced solution you have a true representation of the actual deflection state.

**CLASSICAL SOLUTION**

The deflection of a leaf spring is given by the formula

\[ y = \frac{6PL^3}{Ebnt^3} \]

where,

- \( P \) is the load = 500
- \( L \) is the horizontal length = 36
- \( E \) is the section elastic modulus = 29E6
- \( b \) is the leaf width = 2.5
- \( n \) is the number of leaves = 6
- \( t \) is the leaf thickness = 0.375

From this formula \( Y = 6.102 \). There are several simplifying assumptions in the classical formula, but comparison is reasonable.

Ref: "Practical Stress Analysis in Engineering Design", by Alex Blake

**OCTRAHEDRON - GEODESIC GENERATOR SAMPLE**

This example is intended to illustrate the versatility of the built in geodesic generator.

Radius = 240
Polyhedron type = Octahedron
Frequency = 18; Class = II
Zenith = axis X !!
Breakdown = Method #3

**DOME SETTINGS**

Geodesic fraction = Dome
Dome base is "unleveled" (not necessary for this type)
Geodesic dome is cut at \( Y = 0 \) (on axis Y !!)

**INITIAL SPECS AT GENERATION**

Strut count = 1485
Node count = 514 Plus one center node

**FINAL SPECS AFTER DOOR AND WINDOW MODS**

Strut count = 1475
Node count = 516 Plus one center node

The built in geodesic generator can construct more geodesic types by more methods with more features than any other geodesic generator software available. Although the final model has 1475 struts, there are only 19 different geometric types based on their end-angle attachment and lengths and this includes the door and window frames.

This model is set up for structural analysis with just a simple dead weight condition applied. To understand how to load a geodesic model with the required wind and/or snow loads for professional structural analysis see the "CADRE Analytic white paper" on this subject.
Another specialized geodesic application "CADRE Geo" can generate these basic models as well and will provide the
detail design specs for hubs, struts, and panels including hub connection angles, strut end angles, panel dihedral
angles, etc. as well as categorized like-types of each and generate drawings for mass production.

One can pass models to CADRE Geo from CADRE Pro and vice versa by means 'Utilities/Export text file' and
'Utilities/Import text file'. This way, one can make the doors and windows modifications in CADRE Pro as shown here
and then export the model to CADRE Geo to make use of the additional detail design features in that application.

**OILTANK.FEM (SNOW AND HYDROSTATIC PRESSURE)**

**OIL TANK WITH SNOW AND HYDRAULIC PRESSURE LOADS**

![Diagram of oil tank](image)

The tank is 900 inch diameter and 336 inches high
The spherical dome peak is 144 inches above the rim of the tank.

**MATERIALS**

- Hull wall is 5/8 inch steel plate
- Dome roof is non structural fabric cladding with steel frame (2.5SQ x 0.188) members.
- Tension ring is 1 x 6 inch bar inches
- Wind girder: 1/2 inch steel plate, 3x6 channels for lateral and edge beams, L6x6x1/2 brace

**LOAD CRITERIA**

The tank is filled with oil at specific gravity of 0.8 to a height of 336 inches.
The roof and wind girder are subjected to a 'ground' snow load criterion of 40 psf.

**CONSTRUCTION**

**TOP**

- R = 450 inches
- H = 144 inches
- Both beams and plates
- Rings = 9
- Sectors = 12
- No. Iso adjusted = 4
CADRE Pro Quick Modeler "spheric" dome

HULL

XZ Plane
Plates only
R=450 inches
H=336 inches
Sectors = 60
Sections = 7

CADRE Pro Quick Modeler "Cylinder" template

These two template models are created, saved, and then merged together. (Refer to Help/"Getting started"/Exercise 4).

ELEMENT LIBRARY

Tank hull plate 5/8 steel
Tension ring - Rectangular steel bar 1 x 6 steel bar
Tank top frame - Steel square tube AISC 2.5 SQ x 0.188
Wind girder beams - Channel 3 x 6
Wind girder brace – L 3 x 6 x 0.5
Wind girder - 1/2 inch steel plate

The fabric plate element is given arbitrary low values of E and T, however to mark different snow regions you can set up several copies of this element with different colors (e.g. Central, R1, R2, R3, R4).
GROUP NAMES
To organize output, set these group names.
"HULL" for the cylindrical part of the tank
"R0 TO R4" for the roof plate elements corresponding to slope
"FRAME" for the roof frame members
"TENSION RING" for the tension ring around the tank top edge.

WIND GIRDER CONSTRUCTION
The wind girder is built up at 288 inches above the ground by first making just one 6 degree segment then replicating it around the circumference.

Start by selecting a tank centerline node, then two adjacent hull nodes at the 288 inch level.

Right-click and use Copy nodes relative then choose the radial option with Y as the radial axis. Set 40 inches (width of the girder).
Press **OK** and two new nodes are created outboard at the wind girder edge radius.

Draw 3 new beam elements for one side of the girder segment, one for the left lateral, one for the edge rim and one for the left brace.

In a **counterclockwise fashion** around the horizontal rectangle, select the two nodes at the 288 inch level on the tank and the two new outboard nodes on the wind girder rim. Right-click and use **Insert mesh**. Use **Plate quad mesh – 4 triangles per quad** with Attach all overlapping nodes.
Press **OK** and the plate component of the girder is installed.

Set up appropriate orientation nodes for the beam elements and then set up group names for all the components. (e.g. WGLATERAL, WGRIM, WGPLATE, WGBRACE). It is important to do these details now BEFORE replication!

Replicate the girder segment around the tank by first selecting the tank center node. Then select every element of the girder segment including both beam and plate components.

Use **Utilities/Clone**.

---

**Replicate selected elements**

- **Relative to system**: Global
- **Translational offsets**: 
  - X: 0
  - Y: 0
  - Z: 0
- **Number of replicas**: 59
- **Node naming scheme**: Sequential
- **Attach overlaying nodes**: Unchecked
- **Add Interconnecting beams**: Unchecked

**Rotational offsets**

- **Angle (deg)**: 6
- **Rotational axis**: Y

**Center node**: 476

---

Static samples
Press **OK** to complete the wind girder.

Select, **By group name**, and assign library properties to each of the components in the wind girder.

**RERAINTS**

All nodes on the bottom rim are set to the pinned state.

**DEAD WEIGHT**

The dead weight is developed from the W/L and W/S terms set up in the element library for all assigned elements. Right-click, use **Set loads** then the Dead weight tab. Y negative, Replace, and check ‘Use compensation’.

Using the **Apply dead weight** button yields: -201393 pounds

Open the **File tab** and **Save** the load set as **D.lsb**

**SNOW LOAD**

Snow load pressure is usually accomplished by creating a unit ‘projected’ load set for the relevant groups then creating the snow load case in the load assembler with the pressures associated with each of those groups. In that case, you need no F/A values in the Library and don’t necessarily need more than one entry for the roof.

However, an alternate method is used here by adding separate library definitions for each snow load group and entering an F/A pressure value for each one. This abbreviated method can be used to load the model as long as there are no other items in the element library with non-zero F/A or F/L factors except those that apply to this load condition.
Slopes & Snow (psf)

<table>
<thead>
<tr>
<th>Elements</th>
<th>Slope</th>
<th>Pressure</th>
<th>Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Central</td>
<td>&lt; 20</td>
<td>40 psf</td>
<td>0.277 psi</td>
</tr>
<tr>
<td>Ring 1</td>
<td>21.714</td>
<td>39.14 psf</td>
<td>0.272 psi</td>
</tr>
<tr>
<td>Ring 2</td>
<td>25.648</td>
<td>37.14 psf</td>
<td>0.258 psi</td>
</tr>
<tr>
<td>Ring 3</td>
<td>29.594</td>
<td>35.20 psf</td>
<td>0.244 psi</td>
</tr>
<tr>
<td>Ring 4</td>
<td>33.539</td>
<td>33.23 psf</td>
<td>0.231 psi</td>
</tr>
<tr>
<td>Wind Girder</td>
<td>0</td>
<td>40 psf</td>
<td>0.277 psi</td>
</tr>
</tbody>
</table>

You can click on any plate element in the roof and use the eye viewer to read the slope.

The snow pressures (in psi) are set up for each ring in the element library under F/A and F/L

Fabric Central slope 0 (Set F/A as 0.277 psi)
Fabric R 1 slope 21.714 (Set F/A as 0.272 psi)
Fabric R 2 slope 25.648 (Set F/A as 0.258 psi)
Fabric R 3 slope 29.594 (Set F/A as 0.244 psi)
Fabric R 4 slope 33.539 (Set F/A as 0.231 psi)
Wind Girder slope = 0 (Set F/A as 0.277)

Right-click; use Set loads, then the projected tab. Y negative, Replace existing, Plates only and Check ‘Use compensation’

Using the Apply projected load button yields: -197035 pounds

Open the File tab and Save the load set as S.lsb.

HYDROSTATIC LOAD

Specific gravity of 0.8 is a density of 0.02888 lb/in^3

Select just the tank HULL plate elements.

Right-click, use Set loads then open the Hydraulic tab.
Set Y negative as direction of increasing load.
Set: Replace
Set 336 as the Zero load level
Set Pos (+) and Plates only
Set intensity to 0.02888.
Use the **Apply hydraulic load** button.

Do not save yet!

Notice the loads in the metrics panel.

\[
\begin{align*}
F_x &= -0.000977 \\
F_y &= 2.0516 \times 10^{-10} \\
F_z &= -0.000092 \\
M_x &= -0.023437 \\
M_y &= 1.7223 \times 10^6 \\
M_z &= 0.125
\end{align*}
\]

These should be essentially zero since these are only the pressure loads on the side of the hull which should balance. Notice the significant torsion about the vertical tank axis ($M_y$). This is due to having flat unsymmetrical facets on a cylindrical surface which all are oriented in precisely the same way around the vertical tank axis. With a constant pressure vs. depth these all cancel out but with an increasing hydrostatic load the torque builds up with depth.

The torsional bias can be corrected as follows:

Exit the global loading module with **OK** keeping this hydrostatic load set on the model.

First, select a centerline node (e.g. the bottom center node). Then select all loaded nodes (or just all nodes since unloaded ones will be ignored). It doesn't matter if the pre-selected center node is in this group as well so you can just drag a frame over the whole model to select all the nodes.

Go to **Utilities/Special loading tools** and open **Redirect loads to a point or axis**.

Choose to redirect to an "Axis".

Choose the $Y$ axis which is the tank vertical centerline.

Choose "Relative" or "Away from".

Press **OK**.

Go back to the global loading dialog with **Set loads**

The new loads and moments are:

\[
\begin{align*}
F_x &= -0.000977 \\
F_y &= 0.0 \\
F_z &= 0.000122 \\
M_x &= 0.015625 \\
M_y &= -0.375
\end{align*}
\]
Open the File tab and save the load set as H.1sb.

Since there is no tank bottom, the resultant loads should be essentially zero. If there was a bottom, the downward result would be the weight of the contents.

FULL LOAD CASE

Right-click, use Set loads then open the File tab.

Use the Assemble button. Navigate to the folder with the 3 saved load sets. They should appear on the list along with any other load sets that may be in that folder. You can select and remove any others or just ignore them by making sure their factor is zero.

Set and assumed combined design case condition such as: 1.2D + 1.6S + 1.2H

Apply these factors 1.2 for D, 1.6 for S and 1.2 for H in the Case 01 column. You could add more columns and create many design cases at once.

Use OK to apply the full case load. (Total download = -556927 pounds)

SOLVE

Go to Results/Beam stress maximums

MAXIMUM/MINIMUM NORMAL STRESS, SN1 OR SN2 - MAXIMUM OF EITHER END

Group name Max/Min
FRAME 5.4481E+02 -6.7337E+03
TENSION RING 3.1725E+03 3.0303E+03
WGBRACE 3.5833E+02 -6.3049E+02
WGLATERAL 2.6965E+03 -3.3211E+03
WGRIM 1.0381E+03 9.4085E+02

Go to Results/Plate stress maximums

MAXIMUM/MINIMUM VON MISES STRESS - CENTROID - FRONT SIDE

Group name Max/Min
HULL 6.5850E+03 1.4344E+03
WGPLATE 1.6738E+03 5.3148E+02
PDELTA.FEM (BEAM COLUMN)

This is a beam-column model used to study the advanced analysis solution for beam columns.

This is the subject of exercise 12 in the "Getting Started" in Help.

Solving this model with the Standard (linear) solution will show a 2500 in-lb bending moment. This is shown below using the Load diagram results item and displaying the bending moment:

Beam with axial and lateral load (P-delta) - Help exercise 12
BENDING MOMENT, Z Max = 2500 Min = 0

Now, when solved with the Advanced analysis option using only 1 increment with 100% load, the results show a much larger maximum bending moment of 3437 in-lb. Normally, only 1 increment should be used for beam-column effects.

Beam with axial and lateral load (P-delta) - Help exercise 12
BENDING MOMENT, Z Max = 3437 Min = 0
The ‘Load diagram’ output feature also allows you to overlay multiple result files. Here are both linear and non-linear solutions on the same plot (linear in blue, nonlinear in red).

Beam with axial and lateral load (P-delta) - Help exercise 12
BENDING MOMENT: Z Max = 3437 Min = 0

PLATEELASTFND.FEM (GROUND STRUCTURE INTERACTION)

This sample static model illustrates the application of the ground/structure interaction for a foundation. A dynamic version of this model is also provided with the sample dynamic models and it can be used to solve for vibration modes with and without ground structure interaction influence.

FOUNDATION

W = 300 inches (z direction width for this model)
L = 600 inches (x direction width for this model)
Embedment = 36 inches (depth of foundation below soil surface)

SOIL GEOLOGICAL PROPERTIES

Soil weight density = 0.0553 lb per cubic inch
Soil Poisson ratio = 0.2
Soil shear wave velocity = 3607 inch per sec
Gravity = 386.02 inch per sec squared

\[ G = \frac{\gamma V^2}{g} \]

\[ G = (0.0553)(3607)^2/386.02 = 1863.8 \text{ pound per square inch} \]

APPLY ELASTIC FOUNDATION BOUNDARY CONDITION:

Select the entire foundation's elements, plates only.

Use \textit{Edit/bounds} from the main menu bar.
In the elastic foundation dialog, use the **Footing stiffness** button to access the foundation calculator,

Or from classical foundation formulas you can derive these values.

\[
\begin{align*}
K_x &= 2.533 \times 10^6 \text{ pounds/in} \\
K_y &= 2.667 \times 10^6 \text{ pounds/in} \\
K_z &= 2.689 \times 10^6 \text{ pounds/in}
\end{align*}
\]

And place them into the fields for X, Y, and Z.

Check X, Y, and Z but NOT the normal box

Do NOT check the projected boxes

Press **OK** and the foundation values are parsed to all the nodes of the foundation.

An alternative method would be to allot the spring total values to the 231 foundation nodes.

**LOADS**

The live load is 3500 pounds on each of the upper floor node points and none on the foundation floor.

Dead mass of the foundation and structure is added to this load to create this load condition.
The basic structure and its loading are not sized for reality. They were selected more or less arbitrarily in order to illustrate loading and restraining of surface type elastic foundations which is the subject of this example.

Concrete reinforcement is not considered.

**SOLUTION**

The foundation settles approximately 1/2 inch under live and dead weight.

**POLE.FEM (SIMPLE COLUMN BUCKLING)**

This is a simple column model used to study the advanced analysis solution for instabilities in exercise 11 in Help, “Getting started”.

This is the subject of exercise 11 in the “Getting Started Tutorial”. The tutorial provides several solutions with different configurations of end restraint. Shown here is the pinned-pinned case with the round tube.

Additional entries are included in the Element Library with an angle (L) section using its geometric and also its principal axes for element properties. These can be used for further examination of the buckling features. Step 12 of the exercise.

One can assign a 'reference length' to the beam segments equal to the full member length (100). This is done by selecting the full beam, then, from the Element Editor popup menu use **Set Lengths** and set the segments to a value of 100. Check the setting by selecting the element and using the Eye view tool and looking at the left side of the left panel of the dialog.

In that case, the Euler parameters are based on the 'reference length' and the compression alert will be more sensible for this structure. The Euler values however are still just parameters as they do not include any accounting for the end constraint conditions unless one employs an 'effective' length and assigns it as the segments 'reference length.'

**PRELOADEXAMPLE.FEM (PRELOAD AND THERMAL EFFECTS)**

**PRELOADS AND THERMAL EFFECTS ON BUCKLING**

This is a 60 inch square frame made of 2.25 x 0.125 square steel tubing. The cross bar is a 0.25 inch solid round bar loaded with 100 pounds preload (compression) in the restrained condition. The cross bar is pinned to the frame at both ends.

You can check the 100 pound preload setting by using **Show preload** from the pop-up menu.
Use *Hide preload* to remove the preload label from the diagram.

The maximum stable load in the cross bar per Euler formula is: 14.19 pounds.

Solving with standard solution, the final load (after the preload is relaxed and fed into the structure) in the bar is: 14.85 pounds.

After solving with the advanced solution with 100 increments instability occurs at just over 95% load. Load in the bar at 95% is 14.106 pounds.

Application of the preload solution is equivalent to fixing each end of the cross bar against any horizontal movement and supporting it against any lateral buckling, then applying the 100 pounds of preload (perhaps by a turn-screw mechanism). Then, solving is equivalent to incrementally removing the artificial end restraints of the bar and allowing to structure to seek equilibrium, or buckle, whichever occurs first.

If one removes the 100 pound preload and replaces it with a temperature differential (hotter) of 11.608 degrees Fahrenheit with a temperature coefficient of 6.5E-6 (for steel in degrees Fahrenheit) then the same solution is reached. The rod reaches 14.106 pounds at about 95% of the 11.608 deg temperature application where instability occurs.

Note: a temperature change $T$ is equivalent to a preload by the relation:

$$T = \frac{P}{(AEC)}$$

where $P$ is the preload, $E$ is the elastic modulus and $C$ is the temperature coefficient in degrees per unit length change per unit length.

**PRESTRESSEDCONCRETEBEAM.FEM**

**PRESTRESSED CONCRETE BEAM (EXERCISE 30 IN HELP "GETTING STARTED")**

This exercise is taken from Example 1 of chapter 13 of Gaylord, Forth edition, "Structural Engineering Handbook"

A prestressed-concrete rectangular beam 20 x 30 has a simple span of 24 feet and is loaded by a uniform load of 3000 lb/ft which includes its own weight (See Figure). The prestressing tendon is located as shown [9 inches from the bottom or 6 inches below beam center] and produces an effective prestress [preload] of 360,000 pounds. Compute the fiber stresses in the concrete at the midspan section.

First, assume the beam is supported exactly at the center of concrete section as shown in the diagram.

Supported at the center of section, as shown, the concrete beam can be represented with a standard beam without any eccentric offset. The steel rod system can be represented with an eccentric beam with a 6 inch offset. Later we investigate other options in vertical support location.
STEEL:
The text book problem doesn't specify the steel area and it isn't necessary since the preload is given in force. However, the model must have some specifics to make finite element model so assume:
The tendon consists steel bars: D=1.375 A = 1.485 sq in; I = 0.175; J = 0.351
The 3 bars are spaced at 5.0 inches apart, 6 inches below geometric center of beam

You can represent the tendon system as single element with equivalent properties relative to the 'center of steel' (0, - 6).

E = 29E6
G=1.12E+07
A = 3*1.485 = 4.455 in sq.
Iy = 2(1.485)(5^2)+3(0.275) = 74.771
Iz = 3(0.175) = 0.526
J = 3(0.351) = 1.053

You can set Sy and Sz and St to zero since bending and torsion of the steel aren't important here.
Also, set Ayf and Azf = 1.0

Enter the steel property up as an 'Eccentric beam' type in the element library with offset of Dy = +6.0 inches (distance of support node above the axis of the eccentric beam). You can leave W/L and F/L at zero.

CONCRETE:
The problem doesn't specify the complete concrete properties, etc and it isn't entirely necessary for the approximate solution of the problem. However, the model must have some specifics so let's assume:

Import this from basic shapes as a b=20; d=30 rectangular bar.

E = 2.07E6
G=899000
A=600
Iy=20000
Iz=45000
J=46917.32
Sy = 2000
Sz = 3000
St = 2857.143
Ayf and Azf = 1.0

Set this up as a 'standard beam' type in the element library. You can set W/L and F/L to zero.

DETERMINE THE APPLIED PRELOAD:
Provide a 360 kip effective preload (internal steel stress after elastic concrete relaxation).
Steel to concrete axial stiffness ratio:
(As*Es)/(Ac*Ec) = (600*2.07E6)/(4.44*29E6)/(600*2.07E6) = 0.104014
Augment the effective preload (360000 pound) to account for concrete elastic relaxation.
Applied preload before solution = 360000 (1+0.104014) = 397455
When the assigned preload is applied and solved, the concrete compresses. The preload (397455 pounds) should relax to 360000 pounds.

CONFIRM PRELOAD SETTING:
Set zero external load
Set a vertical Y restraint along the beam allowing for axial displacement only.
Left end: X, Y, Z, Xrot, Yrot FIXED. Zrot FREE
All other nodes: Y fixed; all else free
Select and assign preload in all steel elements at -397455 pounds (negative for tension!)

On solution the final tension (in the steel) should be 360,000 pound confirming the 397455 pound application load.

EXTERNAL LOAD:

Set the beam up as simply supported.

Left end: X, Y, Z, Xrot, Yrot FIXED. Zrot FREE

Right end: Y fixed; all else free

Apply 3000 lb/foot (250 lb/inch on just the concrete beam. Select the concrete beam only and use Set loads. Then open the Projected tab. Set Y direction Negative. Enter intensity of 250. Use the Apply projected load button. Press OK to exit.

SOLVE:

Solve, check stress at the beam center (concrete element!). Select the element just to the left of center.

Component stress (9):

Axial stress = -6.0477E+02
Shear Y = -4.9992E+00
Shear Z = 0.0000E+00
Torsion = 0.0000E+00
Bend My = 0.0000E+00
Bend Mz = 1.3825E+02

Net extreme fiber stress in concrete:

-604.7 + 138.25 = -466 psi

Or,

-604.7 - 138.25 = -743 psi

The Gaylord classical solution shows: -456 psi and -744 psi

The difference is that the offset makes the steel element somewhat more flexible (coupling of axial and bending) and allows somewhat more load to be dumped into the concrete during solution (362860 which is 2860 pounds more than specified). This is a realistic effect and may be more precise.

To get the 'book' solution, reduce the preload by 397455-2860 = 394595

Now the solution gives: -457.43; -743.85 which is essentially the book solution.

Check the axial force in the steel and note that it is now very close to 360000.
ALTERNATE MODEL

If you want to represent the beam as simply supported on its lower side of the section rather than the centroid as shown in the diagram below,

![Diagram of beam with load and support conditions](image)

then, change the concrete property type to the 'Eccentric beam' type and use an offset property of Dy = -15 and reset the steel eccentric beam offset property to Dy = -9.0.

Re-solve: Get precisely the same answers. Vertical support location makes no difference for this configuration and loading case.

POSTSTRESSED BEAMS

See Exercise 30 in Help for details of handling post-stressed concrete beams where dead load must be separated from the other load conditions.

QUONSET.FEM (MULTIFRAME STRUCTURE)

This model is constructed using the sample file Frame.fem.

The sample file Frame.fem is a single complete truss. It is used to build this Quonset hut using the replicating features.

![Diagram of Quonset hut](image)

Start with the frame model Frame.fem.

Make a building 1900 inches long with 100 inch spacing between frames. You will want longitudinal connectors on the nodes that connect each frame. These will support the roof cladding and stabilize the frames. Usually you would strategically include just the amount needed but in this exercise, interconnect at all nodes, internal and external.

REPLICATE THE FRAME

Exit from any editing modes.

Connecting lateral beams can be installed between the replicates at any previously selected nodes. With the selection circles showing, drag the selection frame over the entire frame selecting all the nodes. Then "deselect" the middle ground node by clicking on it once.
Then, go to Utilities/Clone and execute that command.

Set Z = 100 inches for the Z offset value and set X and Y to zero. \(X=0, \ Y=0, \ Z=100\)

This makes the front frame at \(Z = 0\), the second at \(Z = 100\), the third at \(Z = 200\), etc.

Set 19 for the number of copies (the original plus 19 copies will be 20 frames in all with 19 bays at 100 inches each).

Choose the Node naming scheme as **Numeric**.

Check the item **Attach overlapping nodes**.

Check the item **Add interconnecting beams**.

Ensure that the angular rotation offset value on the right side of the dialog is set to zero.

Press **OK** and you should have the completed structure with connectors between each frame.

Since we chose Numeric for the node name scheme (which also applies to the element user names, if any) each frame has a consistent naming system based on the initial name settings with an additional numeric prefix beginning at 1 for the base frame.

So the base frame nodes you started with will now be numbered from 101 to 121 with 122 as the center ground node, the next will go from 201 to 221 with 222 as the center ground node, etc. The last frame will be from 2001 to 2022.

If you are viewing in the XY plane, the initial frame is behind and the last (the 19th copy or frame #20) is out in front since the copies were made to replicate toward the positive Z axis. If you wanted it the other way then you would have used \(Z = -100\) in the clone dialog.

The elements, and also the connectors, have a similar consistent number system as well.

**ASSIGN PROPERTIES TO THE CONNECTORS**

All the frame elements copied their properties and analogous orientations from the base frame, however, the connectors between frames did not exist and don't have any properties or orientation information so you have to provide it manually.

This may seem tedious but there are some shortcuts that can be used.

To assign properties to the connectors it is necessary to select them all without selecting any other elements. There is a trick to selecting all the connectors even though they have not yet been assigned and have no group name.

Go to the Element editor mode and then, under **Select/Elements** on the CADRE Pro menu bar, choose **By group**.

Choose the item "Unnamed". This will select all the elements without a group name which coincidently are the inter-frame connectors for our model since everything else was already given a group name. That is the main advantage to assigning group names as you go along!

Right-click on the screen and assign a group name "Connectors" to these **Unnamed** elements so that you can select them by that name in the future.

Again go to **Select/Elements** on the CADRE Pro menu bar choose **By group**.
Choose the item Connectors again.

With these connectors selected, choose the Properties button and assign the 2.5 x 2.5 x 0.125 x 0.125 properties to them.

You could have accomplished this also using Select/Elements and By type and choosing the Unassigned elements as well.

ASSIGN ORIENTATION TO THE CONNECTORS

In this particular case, any of the nodes along the center at 0,0,0 would serve as a proper orientation node for all of the connectors (e.g. Node 122 at the center ground point of the first frame).

Go to the Element editor.

Then under Select on the menu bar choose Elements and By group.

Choose Connectors, to select all the connectors and then click on the Orient button.

Choose node 122 from the list and Assign this node as a reference for all the selected connectors.

Your completed model should look like the sample model Quonset.fem.

Now is a good time to verify the model again using Tools/Check model/Check for anomalies.

**QUONSET1.FEM (MULTIFRAME STRUCTURE CLAD AND LOADED)**

CLAD AND LOAD A FRAME MODEL

This sample model is created from the sample file Quonset.fem which is an open frame model without cladding.
CLADDING

Using the Quonset.fem model, use Select/Elements/By group and select the inner chords and the diagonal beams. Right-click and use Hide selected elements. This prevents any cladding of the triangular portions of the trusses themselves. You want clad only the outside surface between trusses.

This leaves only the outer chords and connectors. The outer surface in this case is composed of quadrilateral openings for which cladding can be applied. The inner connectors do not contact the outer surface so will not interfere with the cladding process (otherwise you would need to hide them as well).

Enter the Element Editor mode. Choose a central node on the ground and select it as the orienting point that will identify the backside (inside in this case) of the plate cladding to be installed.

Use Utilities/Clad to open the cladding dialog. The selected node will already be shown on the dialog.

Choose to Include quadrilaterals.

Set the maximum intersections to 2 (typically as few as needed but in this case for a 1 dimensional surface it makes no difference).

Choose OK and the cladding is applied.
You can now use View/Elements and Set to show to set all elements back to the visible status.

LOADING CONDITIONS

There are many different ways to apply loading to the external surface. Exercise 10 in help "Getting started" demonstrates in detail how to set up a simple lateral wind case this circular shaped structure.

Those lateral wind loads have been applied to this model.

For simplicity in this demonstration, the surface area is divided with just 3 areas; Windward, Upward, and Leeward. Surface pressures were calculated to be:

- Cladding windward = +0.6373 psi (first three quadrilateral bays on the -X side of the hut)
- Cladding Upward = -0.12518 psi (central area of the roof)
- Cladding Leeward = -0.05690 psi (last three quadrilateral bays on the +X side of the hut)

Select the regions (plate cladding only) one-by-one and give them some group names: Windward, Upward, and Leeward.

Three surface (Kirchhoff plate) property definitions are set up in the Library for each of these regions. Then each region is selected and the appropriate property is assigned.

You can enter the surface pressures for the F/A value for the regions in the Element Library.

Method 1: Simple method for a single loading case

With nothing selected, right-click and use Set loads.
Open the Normal tab.
Choose Negative (to conform to wind loading signing tradition)
Choose Replace existing
Click on the Apply normal load button
OK to exit and the model will be fully loaded with a proper load distribution for a wind to the + X direction.
This method employs the F/A values stored for the regions in the Element Library.

Method 2: More flexible and more intuitive method

Select the windward region plates (Select/By group and choose 'Windward' surface)
Right-click and choose Set loads
Open the Normal tab.
The dialog in this case (with elements selected) provides an intensity field for the pressure entry)
Choose Negative (to conform to wind loading signing tradition)
Choose Replace existing
Enter +0.6373 in the intensity field.
Click on the Apply normal load button

Use the Select button on the Normal tab and choose 'Upward' surface.
Choose Negative (to conform to wind loading signing tradition)
Choose Merge with existing (So as not to remove the windward load!)
Enter -0.12518 in the intensity field
Click on the **Apply normal load** button

Use the **Select** button on the Normal tab and choose the 'Leeward' surface
Choose **Negative** (to conform to wind loading signing tradition)
Choose **Merge with existing** (So as not to remove the windward load!)
Enter -0.05690 in the intensity field
Click on the **Apply normal load** button

This method DOES NOT USE the F/A values stored for the regions in the Element Library.

Method 3: Quickest and most versatile method

You can use the **Unit sets** button on the Normal tab to create a complete set of "unit load files" for the regions (all at the same time) and store them in a folder.

Note: If the **Unit sets** button is not visible, use the **Select** button, choose "uncheck all" and press **OK**. This will place the global loading dialog into the full model mode with the **Unit sets** button visible.

Once all three unit sets are created and stored, you can use the **Assemble** button on the File tab of the loading dialog and navigate to the folder where you saved the unit sets. Then enter the pressures as factors for each of the regions shown on the assembler load set list. You can then save the final load case as set as well as apply it to the model.

This method is best when you have different pressure sets to apply to the same set of regions. In that case you can create and save many loading cases at the same time.

**BOUNDARY RESTRAINTS**

Enter the Nodal Editor mode. Use **Select/Nodes/By group** and choose the 'Ground' nodes. These were identified back in the original Frame.fem model during its construction and carried forward to the Quonset.fem model and finally to this Quonset1.fem model.

Use the **Bound** button on the editor panel, or just right-click and use **Set bounds** from the pop-up menu. On the boundary restraint dialog, use the **Pin** button to restrain these nodes in a pinned fashion to earth.

When finished, use the model checking tools under **Tools/check model** to check for errors.
RECTANGLARDOME.FEM - Rectangular pond cover.

The frame material is aluminum and the rectangle is 20 meter long by 10 meter wide by 4 meter high with an elliptical dome shape fitted to the rectangular foundation.

The chosen consistent units are millimeter for length and Newton for force.

This model is made with the Quick modeler with these specifications

*Rectangular dome template*
- XY plane
- Both plates and beams
- \( L = 20000 \) mm
- \( W = 10000 \) mm
- \( H = 4000 \) mm
- 20 x 10 quadrants
- Elliptical shape
- Right diagonals/Alternating
- geometric grid (i.e. uncheck *Use simple grid*)

Orientations for the frames (lateral) and stringers (longwise) are accomplished with extra nodes placed on the floor automatically by the Quick modeler. The diagonal beams are referenced within the surface adjacent to each diagonal element. The intent with this particular model is to have the stiffest section direction of the frames and stringers normal to the floor plane and the stiffest direction of the diagonals normal to the surface. To this end, the \( I_z \) section property setting for stringers and frames are greater than \( I_y \), but for the diagonals, the \( I_y \) value is set up as the greater one. This setting, in combination with the automatic reference node setup achieves our particular design objective.

You can check the stiffness orientation of beam using the element "Marking feature".

The properties for each element type are set up in the library.

<table>
<thead>
<tr>
<th>Description</th>
<th>Group</th>
<th>E</th>
<th>G</th>
<th>A</th>
<th>I_y</th>
<th>I_z</th>
<th>J</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rect tube 60 X 40 X 3 Base</td>
<td>Base</td>
<td>200000</td>
<td>78700</td>
<td>541</td>
<td>13400</td>
<td>25400</td>
<td>293000</td>
</tr>
<tr>
<td>Rect tube 50 X 30 X 4</td>
<td>S</td>
<td>200000</td>
<td>78700</td>
<td>535</td>
<td>66900</td>
<td>15300</td>
<td>165000</td>
</tr>
<tr>
<td>Rect tube 50 X 30 X 3 Frame</td>
<td>F</td>
<td>200000</td>
<td>78700</td>
<td>421</td>
<td>57000</td>
<td>12800</td>
<td>135000</td>
</tr>
<tr>
<td>Rect tube 25 X 50 X 3 Diagonal</td>
<td>DL,DR</td>
<td>200000</td>
<td>78700</td>
<td>391</td>
<td>11200</td>
<td>36700</td>
<td>96400</td>
</tr>
<tr>
<td>Polycarbonate</td>
<td>Glazing</td>
<td>2206</td>
<td>0.37</td>
<td>12</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Use *Select/Elements/By group* for each group and assign the associated property by description.
LOAD

A representative snow load of 0.000957 Mpa (about 20 psf) is applied to the dome. The pressure is reduced by slope and applied to the dome by selecting panels (By slope) and assigning the appropriate pressures according to the following table.

<table>
<thead>
<tr>
<th>&gt;</th>
<th>&lt;=</th>
<th>P  (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>25</td>
<td>0.000957</td>
</tr>
<tr>
<td>25</td>
<td>45</td>
<td>0.000682</td>
</tr>
<tr>
<td>45</td>
<td>70</td>
<td>0.000372</td>
</tr>
<tr>
<td>70</td>
<td>90</td>
<td>0</td>
</tr>
</tbody>
</table>

The total load should be -140293.84 N

BOUNDARY RESTRAINT

The base nodes have a group name "EDGE" which can be used to select them (By Group) all at once. They are set up as Pinned boundary conditions.

SOLUTION

The highest stress is in the frame structure with the von Mises stress of 44.35 Mpa.

**ROTOR.FEM (Tangential loads)**

Generator rotor - braking action (perhaps a seizure design case)

Diameter = 12 inches

PROPERTIES (all cast steel E=2.875E6, Density = 0.283)

- HUB 1 inch thick from R=0.5 to R=1.0
- Web 0.375 thick from R=1.0 to R=2.5
- Spoke 0.1875 thick from R=2.5 to R=4.0
- Rim 0.500 thick from R=4 to R=6
- 8 spokes 0.75 inch wide

A breaking action brings the 2000 RPM rotor to a stop in 0.2 seconds. Determine the maximum stress in the rotor.
CONSTRUCTION:

QUICK MODELER - SPHERIC
Height: 0
Radius: 6
Rings: 12
Sectors: 8
Boundary shifts: 2

Cut out spokes, then discretize by dividing all by 4 to quadruple fidelity.

The Select/By radius option is the most convenient way to select the concentric regions to assign the properties to the elements.

LOADING

Go to the Global loading dialog.
Open the Rotational tab.
Set Tangential
Set Axis = Z
Press 'Apply rotational load'

This applies The summation of W x r tangentially over all elements where W is taken from the element library.

Read:  \( M_z = 214.331 \text{ in-lb} \) on the dialog metrics panel.

At this point, the load case applied is the rotary inertial load for a value of \( A/g = 1.0 \), where \( A \) is the angular acceleration and \( g \) is the gravitational conversion constant for weight to mass.

FIND MASS ROTATIONAL INERTIA (not needed but just in passing)

\[ I = \frac{W}{g} \times r \] so we can deduce that \( I = \frac{214.331}{386.09} = 0.555 \text{ lb-sec}^2/\text{in} \)

ANGULAR ACCELERATION

2000 RPM = 209.439 radian/sec

Angular acceleration (A) = \( \frac{209.439}{0.2} = 1047.2 \text{ radian/sec}^2 \)

\( A/g = \frac{1047.2}{386.09} = 2.7123 \)

FIND DISTANCE TO STOP (not needed but just in passing)

\[ \theta = \frac{1}{2} \times 1047.2 \times (0.2)^2 = 29.949 \text{ radian or 3 and 1/3 revolutions.} \]

FACTOR UP TO THE TOTAL LOAD

The applied load currently is for an \( A/g \) of 1.0 so we would need to multiple the currently derived load case by \( A/g = 2.7123 \) to get the actual loads for this braking case.

Open the File tab on the global loading dialog and use the Factor button.

Enter the factor 2.7123 in the factoring dialog.

This sets up summation of \((w \times r) \times A/g\) which is the total torque at 1047.2 rad/sec^2

Read \( M_z = 581.33 \) which is the applied torque under this braking action.
SOLUTION:

Use *Result/Rank and color*. Set up Von Mises stress and rank all plates.

Maximum stress is approximately 531.8 psi under the specified condition.

**SBEAM.FEM CONTINUOUS BEAM EXAMPLE**

This is the static model developed in Help 'Getting Started' exercise 2.

100 inch long beam made of a rectangular steel tube 4 x 2 x 0.1875 x 0.1875.

**LOADS**

In addition to its own weight it is loaded with 10 pounds per inch (down).

It also has a discrete point load at the free (left) end.

**RERAINTS**

It has a pivot restraint at node 13 and a fully fixed (embedded) restraint at node 2.

**RESULTS**

The results compared to classical solution (in parentheses) are:

- Displacement (node 1): (-.2434) -0.2437
- Root moment reaction: (8312) 8345
- Root vertical reaction: (-366.2) -366.9
- Bending moment at the support: (23246) 23259
- Support reaction: (1626) 1627

An alternative beam section property is included in the Element Library.

The sample model Tbeam.fem is similar and used as an example in the User Manual to demonstrate combined stress analysis.
**SCISSORS.FEM (SCISSOR CONNECTIONS)**

This model is constructed in help, getting started in exercise 16

This model demonstrates a technique for building a scissors action using pinned beams. This takes some careful modeling around the pivot since beams are fixed (or pinned) to nodes, not directly to each other.

Fortunately there is a Utility called **Insert pivot beam** that handles the pivot construction details.

In this model, the horizontal and vertical beams are pinned at the left wall and floor and free to pivot on the central node (the central node is not bound in any way to earth). A 450 Newton load is applied horizontal at the upper end of the vertical beam or (Node 4) and applied vertically at the right end of the horizontal beam (Node 2).

To create a scissors action, one of the beams (the horizontal one in the sample model) must be set up with offset nodes close to the pivot with another beam element crossing over to the other side through but not connecting to the pivot. The (crossover element) is fixed at both ends. The short segment of beam (pivot arms) between the offset nodes and the center node are made as rigid as practical and are fixed at the offset nodes and free at the center node.

In this model the short stub ends are arbitrarily set with 100 times the stiffness properties of the other beams. When representing 'rigid' members you must be careful to make them only relatively rigid since large stiffness differences between connecting members can lead of numerical significance problems. A couple of orders of magnitude should be enough in this example. A check of the Condition Number often reveals when you have gone too far.

The different elements main beams, crossover beam, and pivot arms are all be set up with different Element Library definitions when using the 'Insert pivot beam' utility so that you can work with them individually. The small pivoting components (crossover and stub beam elements) are overlapping so you have to alternately set the "hidden" property (in the Element Library) of the crossover beam and pivot arms so that you can select, work with, or examine them.

**RESULTS:**

From classical mechanics the displacement at the ends of a hinged beam supported at the mid-span is

\[ \frac{PL^2}{12EI} + \frac{PL}{AE} \] or 28.75 mm.

Results from this model are:

Vertical beam horizontal displacement = 28.77 mm

Horizontal beam vertical displacement = 29.80 mm

These are results with offset of 25 mm for the 2500 mm long beam (2 % bearing offset). If they are reset to 12.5 mm (1 % offset) the displacements are even closer to the classical value.

**SHRINKFIT.FEM (SHAFT SHRINK FIT EXERCISE)**

A 2 inch outer steel shaft is heated to 250 deg F so that it just fits over a 1 inch solid steel shaft.

The object of this analysis is to determine the fit pressure on the 1 inch shaft surface when the outer shaft cools to the same temperature as the inner shaft. The pressure on the surface of the inner shaft is approximately the same as the in-plane stress within the inner shaft elements.
This model is constructed using the Quick Modeler 'spheric' template. Two element definitions are set up in the Element Library mainly to have different colors for inner and outer shaft. The thickness is set at 1 inch unit width.

The temperature is set using the Element Editor, selecting the outer shaft elements, and using the preload button to apply temperature and the expansion coefficient. The inner and outer shaft parts have group names (inner, outer) to facilitate re-selection and setting.

**CLASSICAL ESTIMATE**

\[ P = dE \left( \frac{R_o^2 - R_i^2}{2R_iR_o} \right) \]

where_

- \( d \) = interference (assumed same as the free inner radii movement of the outer shaft when cooled alone.
- \( R_o \) = outer shaft outer radius
- \( R_i \) = outer shaft inner radius or inner shaft outer radius
- \( E \) = Elastic modulus
- \( d = (R_i)(A)(T) = (1)(6.5E-6)(250) = 1.625E-3 \) inch

Where A is the expansion coefficient for cold rolled steel 6.5E-6

Calculate _

\[ P = 17977 \text{ psi} \]

**RESULTS (CADRE PRO)**

S1 = 18023 to 19400 psi on the inner shaft elements adjacent to the surface of the inner shaft.

The principal stresses (S1, S2) will be radial and tangential stresses for a radial-loaded symmetric object.

**SKEWEDTORQUE.FEM**

Loading a skewed shaft with radial, tangential, and torque loads

This sample file for the Help tutorial exercise 32. It is used to illustrate how to apply rotational forces to an unaligned shaft. Three kinds of loading operations are illustrated.

1) Radial centrifugal forces
2) Tangential rotational acceleration forces
3) Shaft torque

This model is in units of Newton for force and millimeter for length.

The consistent mass would be in N-mm/sec2 which would be the metric ton so the 50 kg equivalent would be used as 0.05 tons when mass is used.

The system normal to the shaft is set up by selecting the shaft and using ‘Define system’ from the pop-up menu. Give it a name such as "My plane".

See Help Exercise 32 for details of loading the skewed shaft.
**SLIPJOINT.FEM (SLIP JOINT EXAMPLE)**

This is a model of a jack with side struts supporting with a slip joint.

![Diagram of Slip Joint Example](image)

See Exercise 17 in Help "Getting Started"

**SNAPRING.FEM (SNAP RING)**

This example was taken from a PE exam study guide for mechanical engineers.

![Diagram of Snap Ring Example](image)

It illustrates the use of 'specified displacements' and makes use of the utility for installing arcs and circles and the use of the beam stress option for evaluating beam stresses.

A snap ring is made of 0.177 inch diameter steel wire, bent into an arch of 270 degrees with a mean radius of 1.5 inches.

Find the opening force required to spread the opening by 0.375 inch.

**MODEL CONSTRUCTION**

**File/New**

Set the extents envelop as:

-1.5, -1.5, 0  
1.5, 1.5, 0  
Set Grid = 0.1

In the Nodal editor, turn on the snap-to-grid tool, hold down the Ctrl and Alt keys simultaneously and snap nodes at:

1. 1: -1.5, 0, 0 (just to get it on the screen, it will be reset shortly)
2. 2: 1.5, 0, 0 (this will be reset shortly)
3. 3: 0, 0, 0

Select node 1, right-click and from the pop-up menu use Set coordinate: Enter -1.06066, -1.06066, 0
Select node 2 (only), right-click and use Set coordinate: enter +1.06066, -1.06066, 0

Then, in order, select Node 3 (the center), then Node 1, then Node 2, so that 3 nodes are selected.

Go to Utilities/Insert special structures/Arcs and circles.

Set Segments to 64, choose Arc>180, and check Attach nodes.

Press OK to generate the snap ring arc with the gap between the last two selected nodes.
ELEMENT PROPERTIES

Go to the Element Library. Use the Basic Shapes tool on the library dialog. Set the material by selecting the "Cold Rolled Steel" material to import the material properties. You may need to use File from the Basic Shapes menu and navigate to the material file MatSampIPF.txt which are in the inch-pound-force system of units.

Choose the solid round section type and set \( D = 0.177 \) inch.

Review results of the round bar section:

- Solid Round \( D = 0.177 \)
- Area = 2.46057E-02
- \( I_y \) (Inertia) = 4.81796E-05
- \( I_z \) (Inertia) = 4.81796E-05
- \( J \) (Torsion coefficient) = 9.63592E-05
- \( C_y \) (outer fiber dist.) = 8.85E-02
- \( C_z \) (outer fiber dist.) = 8.85E-02
- \( R_y \) (Radius of gyration) = 4.425E-02
- \( R_z \) (Radius of gyration) = 4.425E-02
- \( S_y = 5.44402E-04 \)
- \( S_z = 5.44402E-04 \)
- weight/length = 6.96342E-03

Click OK to import the basic section properties into the model library.

Set some color so you will get some feedback when the properties are actually applied to the model.

Click OK to leave the library, retaining these changes.

Go to the Element Editor, Select all elements (frame select all at once or use Select/Elements/All) and use Set properties to assign the solid round property to the model.

RERAINTS OR BOUNDARY CONDITIONS

Go to the Nodal editor.

Choose node 1: Fix X, Y, Z, Xrot; all else free
Choose node 2: Fix Y, Z, Xrot; set X to displacement = 0.375; set all else free

LOADS

No external loads need be applied, as loads are the unknown to be found.

SOLUTION

Go to View/Nodes and check the items: Show load values and Show reaction vectors.

Turn on the Load vector tool.

Solve using Static Standard.

EXAMINING RESULTS

Result at the reaction indicates 25.44 pounds needed to spread the snap ring 0.375 inches.

Select node 2 then use the eye view tool. Note that the X displacement is 0.375 inches.

Select node 1 and 2 and use the 'Measure tool' to check the distance between the two points,

- Distance (Initial) = 2.121
- Distance (Displaced) = 2.496
- i.e. net change is 0.375 inches

Use the Rank and color dialog (Results/Rank and color) and with the 'Beam loads option'.

Check the maximum moment, \( M_z \) (Rank on Origin node - Degree Zrot), Get \( M_z \) (max) = 65.119 in lb).
The section modulus $S_z$ from material properties above is $5.44402 \times 10^{-4}$, The maximum component stress due to bending alone would be:

$$M_z/S_z = 65.119/5.44402 \times 10^{-4} = 119615 \text{ psi}$$

However, the stresses can be checked directly by changing the clicking on and changing the output mode to beam stress option.

Rank on Component stress - Choose Degree Zbend: The maximum bending stress is 119615 psi
Rank on Von Mises stress - The maximum Von Mises stress is 120648 psi.
Rank on S1 or Sn1 - Choose Normal (for Sn1): The maximum normal stress is 120648 psi
Rank on Ss: The maximum normal stress is 120648 psi

Rank on Sn1 again. Click on the value 120648 psi in the ranked list to ensure that it is highlighted.

Choose OK to leave the dialog. The model will be stress colored with the last ranking.

Press F12 and the highest stress element (last item clicked) will be highlighted in the selection color.

**STRAIN ENERGY**

Finally, one can often confirm satisfactory results by comparing the strain energy stored in the model to the actual work performed on it. In this case:

$$\text{Work} = \frac{1}{2} \cdot P \cdot d = \frac{1}{2} \times 25.44 \times 0.375 = 4.77 \text{ inch-pound}$$

Under Model/Gross properties (with the model is in the solved state) read the total strain energy as 4.77 inch-pound.

The internal strain energy is equal to external work performed.

**SPIRALSTAIRCASE.FEM (SPIRAL STEEL STAIR CASE)**

Rise = 160 inches with 15 steps and one full circle
Inner radius = 20
Outer radius = 60
Step width = 40 inches
Rail height 28 inches from step heel edge
15 steps to 360 degrees = 24 degrees per step
Riser height = 160/(15+1) = 10.0 inches
Outer step arc width = 2π60/15 = 25.13 inches
Inner step arc width = 2π20/15 = 8.377

ILLUSTRATES
1) Insert mesh
2) Copy nodes relative
3) Constructs; Force to radius
4) Replication, simultaneous translation and rotation

CONSTRUCTION
The procedure is to build just one single step with all of its related components then replicate it.

Start a new model using File/New.
Set extents at -60,0,-60 to 60, 36, 60 with 2 inch grid.

CENTER POLE
Set nodes and (0, 0, 0) and (0, 10, 0) and draw or insert a beam between.

VERTICAL RISER
Set 4 node points in the XY plane
(-60,0,0), (-20,0,0),(-20,10,0),(-60,10,0)
Draw or insert four beam elements around the nodes making a rectangle.

Select the four corners of the rectangle in a counterclockwise fashion (spanning the long dimension first). Right-click and use Insert mesh.

Chose Plate mesh - 2 triangles per quad,
Set: X=8: Y=3 (unless you spanned the short distance first in which case these should be reversed)
Press OK to insert the plate mesh in the rectangle.
Select the upper and lower beam elements and divide by 8.
Select the inner and outer edge beam elements and divide by 3.
Select all nodes and use Attach nodes to tie the frame to the plate mesh.
HORIZONTAL STEP

Replicate the upper edge beam of the riser radially about the center pole in order to create the toe edge of the step.

Specifically, select the 8 upper segments of the riser upper edge beam and use **Utilities/Clone**. Replicate by rotation once (1 copy) at -24 degrees about the Y axis using the center node (0,0,0). Use only rotation, make sure there is zero translation. *(Note: 24 degrees is 1/15 of 360 degrees since the 15 steps complete on full spiral rotation.)*

Draw new inner and outer edge beams on the step just created.

Select and divide both of the new inner and outer edge beams by 4.

Select the 4 corners of the step frame and use **Insert mesh**.
Choose **Plate mesh 1, 8 x 4**, and **Attach all overlapping nodes**.

Now, curve the step edges into arcs...

Select the outer edge **nodes** of the step then the center node at elevation 10.
Use **Utilities/Constructs** then use **Force to radius**, XZ plane, 60 inches.
Repeat for the inner edge nodes using 20 inches.

**NEWELS**

Select the outer corner node on the HEEL edge (riser upper edge) of the step.

Use **Copy node relative** in the Y direction at 28 inches *including a beam connector*;
Select the next adjacent outer edge node and repeat using \(28 + 2.5 = 30.5\)
Select the next adjacent outer edge node and repeat using \(30.5 + 2.5 = 33.0\)
Select the next adjacent outer edge node and repeat using \(33 + 2.5 = 35.5\)
Select the next adjacent outer edge node and repeat using \(35.5 + 10 = 38\)

Perform the same on the inner edge except only on every other node (heel, middle, toe) at 28, 33, and 38.

**HAND RAILS**

Draw beam element between the upper nodes on the inner and outer newels.

Once the rails are finished:
- delete the last inside 38 inch high newel beams.
- delete all eight segments of the toe beam of the step
These deletions avoid duplicates when replicating.

**FINISH**

Apply group names for all components distinguishing step from riser, inside from outside, toe from heel, etc.

Apply proper orientation nodes for all beam elements.

Delete any unused nodes.

Attention to detail before replication saves headaches later!

**STAIRCASE**

Replicate the step model simultaneously in translation and rotation.

Use *Utilities/Clone*.

Set the reference system to ‘Global’

Set translation X = 0, Y = 10 and Z = 0 inches.

Set Rotation about Y axis, -24 degrees; Select Node 1 as the center node (center node 0, 0, 0).

Use 15 replicas (15 additional copies not counting the original)

Naming scheme is *Sequential*.

Check Attach overlaying nodes.
LAST STEP CLEANUP

Only the riser and its newel attachments are needed for the most upper step since the step component is level with the upper level floor at 160 inches. So, delete the upper horizontal step plate elements and all beam elements extending inward beyond the riser but retain the newels extending upward from the top riser as they will be the rail supporting posts on the upper floor.

For structural support some additional bracing to the center pole were also added at 3 equal elevations.

ELEMENT PROPERTY ASSIGNMENT

Representative items for a spiral steel staircase are set up in the library. Using the assigned group names, apply the appropriate properties to each component.

LOAD

The area of a single step is 668.982 square inches. Assume a load of 300 pounds per step on all 15 steps. The pressure per unit area would be 300/668.982 = 0.44844 psi.

Use Set loads. And open the Projected tab.

Check Y negative. Set the Intensity at 0.44844.

Press Apply projected load.

The total load on the stair case is 4500 pounds total.

There are more load cases than one would apply to a stair case including concentrated loads on rails, however, the main objective of this sample is to demonstrate replication with simultaneous translation and rotation.

SPRINGCOMPRESSION.FEM (HELIACAL COMRESSION SPRING)

The displacement of heliacal springs is mainly the accumulative result of torsional displacement of the spring wire.

This example was modeled with the Quick Modeler utility with the Helix template

Length = 2.0 inches (initially)

Diameter = 0.5 inches

Number of turns = 20 (initially)

Segments = 600

The ends were modified by selecting the first coil and setting X to 0.1 for all then selecting the last coil and setting X to 1.9 (i.e Nodal editor; Right-click; choose Set X), creating the collapsed compression coils at the ends. Then select all nodes and use Attach nodes to weld any overlapping nodes together. That leaves only 18 effective spring turns.

Wire section data (derived from the Basic Shapes section property generator):

Solid Round D = 0.05

E=29E6

G=11.5E6

Area = 1.9635E-03

Iy (Inertia) = 3.06796E-07

Iz (Inertia) = 3.06796E-07

J (Torsion coefficient) = 6.13592E-07
Cy (outer fiber dist.) = 2.5E-02
Cz (outer fiber dist.) = 2.5E-02
Ry (Radius of gyration) = 1.25E-02
Rz (Radius of gyration) = 1.25E-02
Sy = 1.22719E-05
Sz = 1.22719E-05
Weight/length = 5.56553E-04

CALCULATED SPRING RATE (CLASSICAL)

Displacement is given by the following equation:

\[ e = \frac{8ND^3}{d^4G} \]

This is the most common formula which assumes all extension is the result of spring element torsion which is essentially true for most springs.

However, there is also a very small contribution from direct shear as well which can be accounted for if desired.

\[ e = \frac{8ND^3}{d^4G} + \frac{4ND}{Gd^2} \]

Using \( d = 0.05 \), \( G = 11.5E6 \), \( D = 0.5 \), \( N = 18 \) (reduced by one flattened turn at each end)
\( e = 0.25043 + 0.00125 = 0.25168 \)

LOADING AND SOLVING

You can carefully select the 30 nodes on the left end and give them a group name such as "Left", and repeat the same on the right end and give them a name such as "Right". Then it will be easier to apply restraints, loads, and examine the results.

Restrain the 30 nodes around the peripheral on the right end as "pinned".

Load the 30 nodes on the left end with a total of 1 pound. (i.e. 0.03333333 pounds per node)

Then Solve/Static/Standard

Check displacement and see an average of ~0.250 inches for the left end movement.

The spring constant would be: \( K = 1/0.250 \sim 4.0 \) pound per inch.

Average displacement of the 30 nodes on the left end is not so readily determined. You can select the left end nodes and use Results/Displacements and try to discern the average X value from the list of 30 nodes. Or cut and paste it to a spread sheet and get the average. An easy method to get it is to check Model/Gross properties and see that the total strain energy after solution is 0.12514 in-lb. You know that the work applied is \( W = \frac{1}{2}(P)(d) \) where \( P = 1.0 \) lb and work applied must be equal to strain energy stored.

So:
\[ E = \frac{1}{2}(1.0)(d) = 0.12514, \text{ and} \]
\[ d = 0.25028 \text{ is the average displacement.} \]

With these pinned reaction points the distribution of reactions will show some spikes due to the tendency for the connect point to lift off (which it can't since it is bound).
To be more realistic and consider the right restraint as hard surface and account for the possibility of lift off you could set the X direction of restraint on the right end at a one way reaction to the left (-).

<table>
<thead>
<tr>
<th>Degree</th>
<th>Restraint type</th>
<th>One way</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Fixed</td>
<td>✓</td>
</tr>
<tr>
<td>Y</td>
<td>Fixed</td>
<td></td>
</tr>
<tr>
<td>Z</td>
<td>Fixed</td>
<td></td>
</tr>
</tbody>
</table>

Or, if the right end surface was considered relatively soft (e.g. soft washer), you can set the X value of restraint to Spring and give it a fairly rigid spring value as compared to the spring rate of the compression spring. This is the way the sample file is configured using 100 pounds per inch for the surface. You could even use the soft spring surface and one-way setting together for the most realistic solution.

ALTERNATIVE SOLUTION USING SPECIFIED DISPLACEMENT

Rather than apply loads, you can remove the loads, set the left end nodes as pinned too, then change the X restraint degree of the left end nodes to "Specified displacement" and its value to 1.0.

Then execute Solve/Static/Standard.

Immediately select the left end nodes (use Select/Nodes/by Group and choose "Left"). Use Result/Reactions and examine the Total value at the bottom of the list of 30 left end node reactions (Rx value). The value is 4.0015 pounds, which is the total reaction on the left end when it is forced to displace precisely 1.0 inches. So K = 4.0015.

STIFFENED PANEL.FEM (STIFFENED PANEL CONSTRUCTION)

This is a 3200mm by 1600 mm stiffened panel made of two skins of 2mm thick cold rolled sheet steel.

The longitudinal stiffeners are five 80 OD x 4 steel tubes. The lateral stiffeners between the longitudinals are angle ("L") 25mm x 25mm x 5mm

LOADS AND BOUNDS

The panel is loaded on the lower "inner" side by a pressure of 0.02 Mpa.

The panel is simply supported along the two short sides.
IDEALIZATION

The tubes stiffeners are idealized as a single line beam elements with short rigid stub "offsets" (40 mm long on each side) where the sheets attach. The lateral panel stiffeners between longitudinals are attached to the sheet but not rigidly welded to the tubes so are treated as pinned ended at the tube intersections.

RESULTS

Von Mises stress:
Sheets >> 176 Mpa
Tube longitudinals >> 135 Mpa
"L" laterals >> 186 Mpa

STORTANK.FEM (STORAGE TANK)

This is the storage tank constructed in exercise 4 in the Help - Getting Started tutorial.

The exercise emphasizes:
- Merging models
- Geodesic generator
- Cylinder template
- Measuring surface slopes (roof slopes)
- Selecting by slope
- Hiding and showing nodes
- Re-orienting the model reference fram
- Applying projected (snow) intensity
- Applying hydrostatic loads
- Applying internal (normal) pressures
- Building up dead weight load sets
- Unit load sets
- Assembling load cases
As shown here, it is loaded only with the snow load in which the pressure varies according to the slope of the roof surface.

The exercise illustrates merging two models (tank hull and tank cover) together to make a single model.

As a practical tank, this model is relatively crude and made coarse and simple in order to concentrate on the objectives of the exercise. In actual practice, the tank wall would be much more finely divided as would the top. A practical and realistic model is `largeconicaltank.fem` also in the sample files and discussed in this document.

The exercise explains how to create dead weight, snow, internal pressure, and hydrostatic pressure loads, and assemble them into combined cases.

See exercise Help “Getting started” exercise 4 for details.

**STRESSCONCENTRATION.FEM (STRESS CONCENTRATION)**

This is a flat plate with a hole under plane stress.

Dimension: 20 inch by 10 inch by 0.125 thick
Hole: 4 inch diameter central hole
Load: 10000 pounds tension force

**CONSTRUCTION**

The model was constructed using a 20 x 10 Plate Mesh from the Quick Modeler and a matching 6 x 6 x 4 inch Hole Mesh from the Hole mesh utility. A 6 x 6 section was cut from the plate and the hole mesh model was merged in and attached. Some of the triangular elements around the hole were strategically subdivided using the Divide (by 2) option.

The in-plane NOMINAL tension stress is 8000 psi nominal based on full section area.

**THEORETICAL RESULT**

Stress concentration factor = 3.72 (Roark, *Formulas for Stress and Strain*)

\[ S_{\text{max}} = 3.72 \times 8000 = 29760 \text{ psi at the top and bottom edge of the hole.} \]

**SOLVE**

Rank and Color using the S2 principal stress

Try viewing the color (F2) with the "hide plate boundaries" item checked in *Options/Settings, Plates tab.*
Try viewing the color (F2) with the "Relative" option in **Options/Settings, Colorize tab** (color range)

**CADRE PRO RESULT**

S2 max = 26430 psi

Concentration factor =3.3 (~ 89% of theoretical)

Notice the narrowing (necking) of the plate width in the area of the hole.

The difference is caused by discretization. The stress in a "plane stress" plate element due to the in-plane stress (as opposed to bending stress) is calculated as the average over the triangular plate so it makes no difference where the stress is read (centroid, mid-side, or vertex). But in reality, the actual maximum stress is on the boundary of the hole and drops rapidly over the area of the first adjacent element. So the maximum "in-plane" stress will always be underestimated near concentrations for models like this.

If the plate is divided one time using the **Divide by 4** option (4 times as many elements) then the factor rises to 3.57 and so on until approximately 3.72 is reached.

Bending stresses (in contrast to in-plane stresses) are calculated according to user designation of centroid, mid-side, or vertex and so a better estimate with fewer elements is possible under plate bending situations.

See the sample file 'holes.fem' for more other ways to make holes in plates and meshes.

**STRUTLFT.FEM (LIFTING STRUT SYSTEM)**

This is the lifting strut model developed in exercise 3 of "Getting started" in Help.

To best view this model turn on the node names, node circles, the load arrows and the extents envelope all with tools on the tool bar. Then move it to obtain a reasonable perspective view.

The nodes have custom names that correspond to the instructions in exercise 3.

**STUDWALL.FEM (STUD STIFFENED WOODEN WALL)**

This is a wooden wall made of standard 2 x 6 studs placed 16 inches on center with plywood sheathing.

The studs are made of eccentric beams (standard style) to account for the fact that the sheathing is not located at the stud centroid. The sheathing, in fact, lies on the outside of the stud arrangement.

In this particular example a simple concentrated vertical load (-5000 pounds) is applied at the top of the center stud and an out of plane load (-1000 pounds) is applied to the center of the stud. This is the kind of simple load scheme one would apply in order to check the integrity of the model.

For example make a check of the strain energy.
Solve and look under *Model/Gross properties* and see that the total energy displayed is 84.29 in-pounds. This is the sum of all the internal strain energy of all the elements in the model under their displaced state.

Next calculate the work applied to the model by the external loads.

Displacement under the 5000 pound load is 0.01311 inches; work is \( \frac{1}{2}(5000)(0.01311) = 32.78 \text{ in-lb} \)

Displacement under the 1000 pound load is 0.103 inches; work is \( \frac{1}{2}(1000)(0.103) = 51.5 \text{ in-lb} \)

Total work done = 32.78 + 51.5 = 84.28 in-lb

Compare to total strain energy stored in the solved model; 84.29 in-lb

Energy is one of the many checks that can be conducted on a newly fabricated or modified model.

Solve and use *Rank and Color*.

Check and show the von Mises stress in the sheathing; 593.6 psi

Check and show the von Mises stress in the studs; 1001 psi.

**TANKHULL.FEM**

This is the cylindrical portion of the storage tank used in constructing *Stortank.fem* by merging two models (*TankHull.fem* and *TankTop.fem*) together in Help "Getting started" exercise 4. The main use of the tank hull and its top are to practice using the merge operation to join two models.

The tank is constructed with the Quick Modeler cylinder template with these settings:

Template: Cylinder
Set plane: XZ plane
Set X=0, Y=0, Z=0
Radius = 400 inches
Height (Length) = 450 inches
Eccentricity = 1.0 (round)
Sections 5 (each 90 inches high)
Sectors 20 (each 18 degrees of arc or 125.66 inches wide)
Plates only: Select
Iso triangles: Un-checked
**TANKTOP.FEM**

This is the elliptical geodesic top of the storage tank used in constructing Stortank.fem by merging two models (TankHull.fem and TankTop.fem) together in Help "Getting started" exercise 4. The main use of the tank hull and its top are to practice using the merge operation to join two models.

This file is created by using the geodesic generator with the following parameters:

- Frequency = 4
- Polyhedron type = Icosahedron
- Radius = 100
- Eccentricity = 0.5
- Fraction = Dome
- Modify base = Unchecked
- Class = I
- Zenith = Y
- Output = use Both plates and beams

There are 20 nodes on the rim, matching the tank hull rim nodes.

**TANKINERTIA.FEM (HYDROSTATIC INERTIA LOADING)**

The tank is 2000 mm square and 1000 mm deep. It is supported simply at the bottom four corners. It is completely full of water and subjected to inertia loading of 1.0 g down and 0.75 g lateral (parallel to one side).

Units are in Newton and millimeters.

**CONSTRUCTION:**

This example is constructed in a fairly coarse manner in order to illustrate the loading method. Typically one would use a finer mesh for this type of model.

Go to Utilities/Quick modeler and choose the Box template.

Length: 2000
Width: 1000
Depth: 2000
Length divisions: 10
Width divisions: 5
Depth divisions: 10
Closed top
Isosceles triangles

The acceleration direction would be slanted by ArcTan(.75/1.0) or 36.87 degrees

**DEFINED SYSTEM FOR LOADING PURPOSES**

Create a defined system oriented with respect to the resulting forces.
Working on the front side of the tank, select the upper right corner and use 'Copy nodes relative' to place a node 1500 units below the top surface (i.e. 2000 Tan(36.87) = 1500).

Still working on the front side, select the upper right node, then the upper left node, then the new point at 1500 units below, and use Define system. Name the system 1D 75L Parallel for reference.

Assuming the tank is entirely full of water. The zero load level with respect to the defined system would be +1200 millimeters above the system XZ plane in the Y direction (i.e. 1500 Cos (36.87) = 1200 millimeters. (See the diagram above)

The resultant load factor is:

$$\sqrt{1^2 + 0.75^2} = 1.25g$$

The rate of load intensity based on the density of water (9.8E-6 N/mm^3) in is:

Intensity = $1.25g \times 9.8 \times 10^{-6} = 1.225 \times 10^{-5}$ Newton per square millimeter per millimeter.

This is the rate of change of force and it increases in the direction of the resultant force (- Y' relative to the defined system).

The intensity of load based on the density of water in Newton per square millimeter per millimeter is $1.225 \times 10^{-5}$ N/mm^3.

This is the rate of increasing pressure and it increases in the direction of the resultant force (- Y' relative to the defined system).

Select all elements (assuming the tank is entirely full).

Use Set loads.

Open the Hydrostatic tab.

Choose Plates only.

Choose the 'Defined system' "1D 75L Parallel".

For the direction of increasing load, Check 'Respect to defined system'

Choose Y', Negative. (The prime symbol indicates relative to the defined system rather than relative to the global system).

The zero load level (with respect to the defined system) is at Y' = +1200 millimeters above the defined system origin per the diagram. Use Pos (+) as the loads to retain.

Enter 1200 for the zero load level.

Choose Replace

Use compensation (compensation is better for this type of mesh).
Enter the load intensity: 1.225E-5.

Use **Apply Hydrostatic load**. Press **OK** to exit.

**LATERAL DIAGONAL INERTIA**

If the load is to be applied laterally across the diagonal and combined with 1g down then you would create a similar system across the diagonal. The upper right back node would be copied to 2828.42 Tan (36.87) = 2121.3 mm downward to create the new system X axis pointer.

Then select the upper-right-back corner node, the left-front-upper corner node, and the new X axis pointer node, then right-click and use **Define systems**.

Name the system "1D 75L Diagonal" for reference.
You can work out the zero load level simply by geometry or by just taking measurements on the screen. You can measure distances between any two selected nodes and angles between 3 of them.

Once you have the necessary measurements you can simply determine the perpendicular distance from the system X axis and the peak node which is the top of the fluid where the pressure is zero.

\[(2828.4 \times \sin(36.87)) = 1697.04\]

So the load case would be applied in exactly the same way as the parallel system but with the new diagonal system selected and the zero load level set to \(Y' = +1697.4\).
Use Apply hydrostatic load.

PARTIALLY FULL TANK

The linear loading function will continue to load elements above the zero load level but in the opposite direction if they are selected. If the tank isn’t full you have to work out the plane of the location of the surface by geometry in order to establish the zero load level.

Often, for a complex geometry tank, the zero load level for an oblique inertia is difficult to determine. One way is to avoid the geometric calculation is to use a simple trial and error approach.

Here is how you would work out a 40% full solution oriented on the oblique diagonal direction 1D 75L Diagonal shown above.

1) Calculate the percentage full desired in terms of fuel weight. 49000 x 0.4 = 19600 N
2) Select the entire external surface and proceed just as described above using the diagonal defined system
3) Set the hydrostatic tab up precisely as before with the diagonal system "1D 75L Diagonal"
4) Be sure to use the Pos (+) option. Estimate the Zero load level starting high.
5) Use the Apply Hydrostatic load button and note the resultant load on the metrics panel and compare to 19600 N
6) Revise the zero load estimate and repeat, closing in on the zero load level that gives approximately 19600 N
This example demonstrates buckling analysis of tapered columns. The section sizes were arbitrarily chosen so that the ratio of the small to large section would match an exact table value in a reference text for comparison of results.

Aluminum 6061 round bar (E = 1.0E7)

\[ D_1 = 1.12468 \text{ inch OD} \]
\[ D_2 = 2.00 \text{ inch OD} \]
\[ I_1 = 0.07854 \text{ in}^4 \]
\[ I_2 = 0.7854 \text{ in}^4 \]


Timoshenko gives the formula \( P_{CR} = KEI_2/L^2 \)

where \( K \) is a function of the ratio \( I_1/I_2 \)

\[ I_1/I_2 = 0.1 \]

At \( I_1/I_2 = 0.1 \) >> \( K = 4.81 \) (for symmetric solid prismatic bar with linear double taper)

\[ P_{CR} = 3777.8 \text{ pound} \]

**MODEL CONSTRUCTION**

(1) Create in the Quick Modeler

End node 1: (0, 0, 0)

End node 2: (100, 0, 0)

Number of segments = 2

For local system orientation node use (0, 10, 0)

(2) Set the directional orientation

Select just the right hand segment of the new beam and use *Tools/Switch elements nodes*.

Now both segments are set with their end nodes as the origin and their axis nodes at the middle (node 4).

This allows us to set a single taper definition to both the left and right segments.

(3) Set definitions in the Element Library

Import from **Basic Shapes**

Material type set to Aluminum 6061 (Use the material file *MatSampIPF.txt*)

Import a "Round bar at 2.0 inches OD"

Repeat and

Import "Round bar at 1.12468 inches OD"

Also, on the Element Library dialog.

Use the **Add** button to to create a third entry section and set its type to "Tapered beam"

Set its description to "TB"
Select the the 1.12468 OD entry on the list and use F8 key to copy
Click on the tapered beam (TB), use F9 to paste (Choose "Origin end")
Select the the 2.0 OD entry and use F8 to copy it
Click on the tapered beam again; use F9 to paste (Choose "Axis end")
Exit the Element Library with OK.

(4) Assign the properties
Go to the Element Editor mode
Select both elements of the model
use the 'Property' button on the Editor panel and assign "TB" to the two elements.

(5) Boundary conditions
Restraints are set as:
Left end node 1: Free, Fix, Fix, Fix, Free, Free
Right end node 2: Fix, Fix, Fix, Fix, Free, Free

(6) Load condition set somewhere above expected result
Set 5000 pounds for Fx on Node 1

SOLVING

CADRE Pro ADVANCED solution (with 50 increments) 48% load >> \( P_{CR} = 2400 \) pounds
With only two segments the result is excessively conservative.
Remove these results (Results/Remove results)
Select both segments, right-click on the screen, and use 'Divide elements'. Divide into 8 sections.
This automatically tapers each segment appropriately and adds corresponding sections to the Element Library and they are automatically assigned correctly to the new model segments (TB-1, TB-2, etc.).

CADRE Pro ADVANCED solution (with 50 increments) 74% load >> \( P_{CR} = 3700 \) pounds

CONCLUSION

1) Tapered members should always be subdivided between main structural connections for accurate stability prediction.
2) As fidelity increases, the prediction approaches the precise value from the conservative side. The prediction is never unconservative.

AMENITIES

Drawing widths and colors can be set up in the library after the fact presentation effects and are not essential to solution. The apparent block shape is because only one width is allowed per defined element. The defined tapered elements are tapered mathematically.
**TBEAM.FEM (BEAM STRESS ANALYSIS)**

This is the example used in Chapter 15 of the User manual to provide instruction in the handling the stress analysis of sections with a biased centroid (e.g. T, L).

100 inch long beam made of a T beam section.

**SECTION PROPERTIES**

T-section 4 x 4 x 0.375 x 0.375 x 0.1875
Area = 2.87446E+00
Iy (Inertia) = 2.01674E+00
Iz (Inertia) = 4.36646E+00
J (Torsion coefficient) = 1.47827E-01
Ry (Radius of gyration) = 8.3762E-01
Rz (Radius of gyration) = 1.2325E+00
Ct to edge on Y = 2.86547E+00
Ct to edge on Z = 2.0E+00
Sy = 1.00837E+00
Sz = 1.52382E+00
St = 0.0E+00
Weight/length = 8.13473E-01
R fillet = 1.875E-01

Additional "near side" modulus values:
Ct to edge on Y = 1.13453E+00
Ct to edge on Z = 2.0E+00
Sy = 1.00837E+00
Sz = 3.84868E+00

A "T" beam has a displaced centroid which causes some special issues in interpreting the correct result since there are multiple locations where the stress can be read with greatly differing values.

If one uses the "near side" values then the envelop J H K N M L will be where stresses can be determined and the important points would be J H and K.

If one uses the "far side" values then the envelop B A C F D E will be where stresses can be determined and the important point would be A at the stem tip.
The Basic Shapes module allows one to opt for near side or far side values when importing properties for basic biased sections like the T and L.

Point A or H can be examined when the lateral section modulus Sy is set to zero with far edge and near edge properties respectively.

Five sets of section properties are set up in the library, differing only by the values of Sy and Sz.

- **T-Far edge**
- **T-Near edge**
- **T-Far edge with Sy = 0**
- **T-Near edge with Sy = 0**
- **T-either with Sz = 0**

### HAND CALCULATIONS FOR COMPARISON

**Point A (stem tip)**

Located at CZ = 0 on the centerline and CY = -2.86547 below the centroid

\[
Sn = \frac{FX}{A} + \frac{MY}{IY} + \frac{MZ}{SZ}
\]

\[
= \frac{-2000}{2.87446} + \frac{10000(0)}{2.01674} + \frac{10000(-2.86547)}{4.36646} = 7258.23 \text{ psi}
\]

**Point H (cap center)**

Located at CZ = 0 on the centerline and CY = +1.13453 above the centroid

\[
Sn = \frac{FX}{A} + \frac{MY}{IY} + \frac{MZ}{SZ}
\]

\[
= \frac{-2000}{2.87446} + \frac{10000(0)}{2.01674} + \frac{10000(1.13453)}{4.36646} = 1902.48 \text{ psi}
\]

**Point J (cap flange tip back side)**

Located at CZ = -2 and CY = +1.13453 above the centroid

\[
Sn = \frac{FX}{A} + \frac{MY}{IY} + \frac{MZ}{SZ}
\]

\[
= \frac{-2000}{2.87446} - \frac{10000(-2)}{2.01674} + \frac{10000(1.13453)}{4.36646} = 11819.49 \text{ psi}
\]

**Point K (cap flange tip front side)**

Located at CZ = +2 and CY = +1.13453 above the centroid

\[
Sn = \frac{FX}{A} + \frac{MY}{IY} + \frac{MZ}{SZ}
\]

\[
= \frac{-2000}{2.87446} - \frac{10000(2)}{2.01674} + \frac{10000(1.13453)}{4.36646} = 8014.49 \text{ psi}
\]

### READING STRESSES FOR BIASED SECTIONS

For each case, a different property set as noted is assigned.

Examine normal stress cases at axis node (13) for element S12*13 under these different property sets.

**T-Far edge properties**

- \( Sn_1 = -1.718E + 04 = B \) (off the section)
- \( Sn_2 = 1.578E + 04 = F \) (off the section)

**T-Near edge properties**

- \( Sn_1 = -1.321E + 04 = N \) (off the section)
- \( Sn_2 = 1.182E + 04 = J \) (cap flange edge)

**T-Far edge, Sy = 0 properties**

- \( Sn_1 = -7.258E + 03 = A \) (stem tip)
- \( Sn_2 = 5.867E + 03 = D \) (off the section)

**T-Near edge, Sy = 0 properties**

- \( Sn_1 = -3.294E + 03 = M \) (off the section)
- \( Sn_2 = 1.903E + 03 = H \) (cap center)

**T-either with Sz = 0 (near and far edge are the same in this case)**

- \( Sn_1 = -1.061E + 04 = Q \) (off the section)
- \( Sn_2 = 9.221E + 03 = P \) (off the section)
Typically for a symmetric section such as a tube, H-beam, rectangle, solid rounds, bars, etc one need not be concerned with the location of the stress as it will always be on one of the corners. However, with sections with a biased centroid (T, L), care must be taken to ensure that the stress is read at a point of interest on the section and not off in space.

For more detail on this example, see the article in Chapter 15 of the CADRE Pro user manual.

**THERMALSTRIP.FEM (BI-METALLIC THERMAL STRIP)**

Thermal strip made of strips (4mm x 1 cm x 10 cm long) of titanium and copper.

The model includes a steel frame and a set of bimetallic strips (Upper is titanium, lower is copper) bonded together. Contact points are located at the right end.

There is a line element drawn between the contact points which doesn't do anything structural but provides one of several ways to checking clearance before and after solution. The bimetallic strips are each made of "eccentric beam finite elements" they are 4 mm thick each so the total strip thickness is 8 mm. The 'eccentric beams' are shown 2 mm offset since that is the distance from the center of the pair to the neutral axis of a single strip. The contacts are initially set with a 4 mm gap between nodes 3 and 4.

**RERAINTS**

Node 1 is fixed in all degrees of freedom, all else is free.

**LOADS**

There are no external loads, and no preloads. Only a temperature change of 100 degrees Celsius is applied. The temperature is applied to the entire device including the frame, bimetallic strips, and contact elements.

**SOLVE**

Use **Solve/static/standard**

The output is the vertical displacement at the right end of the beam and the main item is the length change of the 'Line' element that indicates the gap. The initial screen displacement is exaggerated and does not indicate a true displacement.

Go to the exaggerate control and set it to zero. Then select the gap line element and use the view tool.

The initial length is shown on the left panel as 4mm. The final gap is read from the right panel under "Loaded length" as 1.863 mm. So with a 100 degree Celsius temperature change the 4 mm gap remains open at 1.863 mm. (Note: Length only shows if "Beam Length" is checked on the Options/Settings/Results tab for the 'Quick viewer' display and 'Results' display)

Use **Solve/static/advanced**

If you want a true view of the displacement under solution then use the **Solve/Static/Advanced** method with one increment. Leave the item "Plot with true displacement" checked. The results are the same but the initial presentation is not exaggerated (until you use **Reset**). Actual results are the same.

All parts of the model expand including the frame and contacts since they are heated as well. For example, point 4 on the frame moves by

\[ X = 0.1170 \]
\[ Y = 0.0094 \]

One can also examine the gap by selecting nodes 3 and 4 and using the measuring feature. To measure the gap, right-click and choose 'measure' from the pop-up menu or use the corresponding tool. In the solved mode, both the initial and final separation between the selected nodes 3 and 4 is displayed.
CONSTRUCTION

One way to build such matching structures with overlaying eccentric beams is first to build one simple beam at 100 mm with the required number of segments. Then, put both offset beam element types in the Library. Then, select and assign one of the properties. The beam will then offset the required amount for that assigned element enabling easy access to draw the other beam. Next, draw the second beam element with the element editor across all nodes but this time draw in the opposite direction (so that the element identification numbers will be different. Then select these new elements and assign the opposite matching offset beam. Using group names and strategic use of the hide and show features can also be useful when dealing with overlaying structures.

DUPLICATE ELEMENTS

Back-to-back eccentric elements will be detected as duplicates by the Tools feature Delete duplicate elements unless they have different offset values (as they do here). So it is best use the same reference node direction for both strips and use opposite signs for the offset values to ensure that they are detected as different beams and not duplicates.

UTILITYTRAILER.FEM (Small utility trailer)

This trailer is designed to carry its own dead weight and an additional 500 lb evenly spread over the bed.

The trailer has a slight up-tilt toward the rear so that with a typical load it settles to a more level position. Actually the initial construction was level, then the feature Utilities/Constructs/Rotate was used to rotate the model about its tongue node a few degrees upward. The same feature is used again to rotate only the undercarriage components (about the leaf spring forward node point) back down toward ground level creating the model you see here.

The wheels are non-structural decor and useful mainly for reference. The axle hub is restrained against vertical and lateral movement. The tongue node is a pinned reaction in all three translational directions.

The main objective of this exercise was to demonstrate the installation of leaf springs and the use of pinned beams to create the articulated axle support system. Pinned articulation requires careful consideration of orientation and pin direction choices. The use of the feature Show pins is essential for verifying the pin orientations.

One must exercise caution with articulated members since the analysis assumes linear with small displacements so significant geometry change could invalidate the results.

Check the ‘True displacement’ check box to evaluate the geometry change under load. Otherwise the displacement is exaggerated to the percentage shown.
It you want to observe the behavior under load, change the exaggerate value to zero then check and uncheck the check box repeatedly.

The leaf spring is the subject of another sample file; Leafspring.fem. That one is a half model of a single leaf spring. It was merged, replicated, and flipped to create a complete leaf spring. Then it is replicated to the opposite side.

The articulated strut support system and even the springs themselves are probably over designed for a small utility trailer but serve to illustrate the use of these features in CADRE Pro.

LOADING THE TRAILER

Use Set loads with nothing selected. Choose the Dead weight tab.

Set Y; Negative; Check Use Compensation;

Click the Apply dead weight button

Open the File tab and save the dead weight load file (-678.44 lb) using the Save button.

Select only the bed plate elements. Choose the Normal tab.

Set Negative; Plates only; Check Use compensation

Enter the load intensity = 500/5520 = 0.09057971 lb/sq. in.

Click the Apply normal load button

Open the File tab and save the payload file (-499.91 lb) using the Save button.

(Note: in this particular case you could also have used the Dead weight tab with Y negative since the payload is a gravity type load or you could even use the Projected tab with Y negative since the bed is essentially horizontal)

On the File tab, use the Merge button to merge the saved dead weight file with the existing payload on the model.

This is the gross load.

Save the gross load file in case you need it later (-1178.349).

Use OK to exit the global loading dialog

Often you want may want to offload all intermediate panel forces that are not supported by frame elements and apply load only to the frame element nodes, especially if the panels are thin or non- structural.

Do this by using Utilities/Special loading tools/"Move panel loads to frame". Now, the trailer frame is now fully loaded (with -1178.349 lb) only on frame supported nodes, and ready to solve.

SOLUTION RESULTS

With maximum load the trailer stern sinks 3.5 inches from an initial height of 13.8 inches or to 10.3 inches. The tongue node is at 11 inches so it settles to 0.7 inches below the tongue node at its maximum design gross weight.

The trailer structural members are not stressed significantly under this load condition. The maximum stress, as would be expected is in the leaf springs and is on the order of 32000 psi.
This example makes use of the Quick modeler utilities ("Spheric" to make the roof and the conical floor) and the "Cylinder" model to make the main barrel. The Merge utility is used to join the three components. The model is loaded with dead weight and a hydrostatic load to the top of the cylinder portion.

Geometry:
- Volume below roof about 54465 ft$^3$
- The cylindrical portion is 20 feet tall by 16 feet diameter
- The conical floor drops 32 inches at the center
- The 5 legs are 9 feet (108 inches) long
- The spherical roof rises 32 inches at the center

CONSTRUCTION

Cylindrical tank component:
Quick modeler - Cylinder; XZ plane; Plates only; R=96; L=240; Sectors = 40; Sections = 20

Spherical roof component:
Quick modeler - Spherical template; XY plane; R=96; H=32; Rings = 8; Sectors = 5

Conical floor:
The profile sketch of the floor is shown below. I would drop 32 inches from the outer flat flange to the conical point but that point is flattened at the radius of 12 inches. (The cone height would be 96 x 32/84 = 36.5714)

1) One method using the Shape surface utility on a flat floor model:
Quick modeler - Spheric template; XY plane; R=96; H=0; Rings = 8; Sectors = 5
Then, select the center node and also select all the plate elements
Use Utilities/Constructs/Shape surface - Polar power equation; Slope A=32/84 = 0.380952381; M=1; W=Z

2) Another method is to generate the Quick model cone (alternate) template.
Quick modeler - Cone (alternate) template; XY plane; R=96; H=-36.5714; Rings = 8; Sectors = 5
Flatten the edge: Using the Nodal editor, select one node at the first ring inside the rim, then select all the rim nodes as well. Use Set Y and set to the displayed default (level of first node selected).

Flatten the cone apex: Using the Nodal editor, select one node at the first ring from the cone apex, then select the cone apex node as well. Use Set Y and set to the displayed default (level of first node selected).

At the flat ring portion of the floor around the rim at each of the 5 pentagon radials, rigid connector beams are inserted between the outermost node and the adjacent radial node. These are divided in half to create a node at the center where the 5 legs will extend 9 feet.

The three components are saved individually and then merged as a single model using Utilities/Merge.
Working on the full model, set the top above the floor to Hidden aid the leg construction process.
Select the 5 center nodes.

Use **Copy nodes relative** with the option to *include extension beams*. Extend to 108 inches.

The leg cross beams and diagonals are created in typical construction fashion. Two bays have X bracing, two have a single diagonal, and one is open to allow equipment entry.

The entire model is reoriented with Y vertical, then translated until the bottom of the legs are at Y=0 to be consistent with the following loading specification.

**LOADS**

The load condition is total dead weight plus hydrostatic loads from a full tank of water.

Select all the plate elements below the spherical roof (cylinder and floor). Choose **Set loads**; Hydrostatic tab

Use **Replace**; Direction of Increasing load is *Y-negative*; Zero load level is 348 (top of the cylindrical portion). Intensity is 0.0361 (i.e. lb/in² per each inch of depth or just density for water in lb/in³). Apply the load, and then use the File tab to save that component. Use **Exit**.

Select nothing, the use **Set loads**. Open the Dead weight tab. Set *Y negative*; Replace; and then use **Apply dead weight**. Open the File tab to save that component as well. Then, on the same File tab, use **Merge** and merge the hydrostatic load to the dead weight load. You could also save the resulting set as the total load.

**RESTRAINTS**

The 5 legs are pinned to ground.

**WATERTANK.FEM (WATER TANK)**

To see the hydrostatic load distribution in the tank, turn on the Load Vectors tool.

**SPECIFICATIONS**

Units: metric Mpa system
Volume is 1204.3 cubic meters
Water density at 20 C is 9789 Newton/M³ or 9.789E-6 Newton/mm³
Water pressure varies from top to bottom of the tank where it is 17000 mm deep

MATERIALS
Legs are Universal Columns UC 254 x 254 x 107
Stiffeners are Square tube 150 X 10 eccentric offset at 75 mm
Hoops and vertical stiffeners integrated with the tank wall are modeled as "eccentric beams" to consider the effects of offsets from the neutral axes when used back-to-back with the tank wall.
Upper hoop is a Square tube 150 x 10
Lower hoop is a Universal Beam UB 254x102x28
Braces are Square tube 150 X 10
Tank wall 10 mm structural steel, 12 meter cylindrical + 5 meter radius hemisphere.

TANK CONSTRUCTION
The tank was made by starting with a single composite curve consisting of a vertical 12 meter portion and a quarter arc 5 meter drawing made with the Utilities/Insert special structures feature.
The arc was discretized at 16 segments and the vertical line at 24 segments to create 40 levels total.
This line was cloned rotationally 9 degrees 40 times around the Y axis with all nodes pre-selected.

The option to *Add interconnecting beams* at selected nodes was checked and this results in a quadrilateral wire frame.

The start and end boundary of the replication line will be overlaying, creating duplicate elements. That must be corrected by using the operation *Tools/Delete duplicate elements*.

The wire frame is then clad using the quadrilateral option. Some point with an inside view of all the interior surface of the tank should be identified and used as the ‘backside’ orientation for the plate elements.
Once clad, the wire frame can be selected and deleted. However, for the final model you can keep the upper rim (hoop) beam, the lower rim (hoop) beam at the bottom of the straight section, and the verticals at 4 locations since they can be configured as the hoop and vertical stiffeners in the final model.

Some improvements were made to the bottom portion of the tank to get better aspect ratio for the elements.

A circular portion was cut out (Radius 1451.4 and elevation of 215.3) and a new patch made with the Quick Modeler “Spheric” (6 x 8 x 1 with 40 supports). The size (radius and height) are not important since the merge operation will scale to fit. However the ratio of radius to height must provide a spherical radius of 5000 mm to match.
The spherical radius is displayed on the dialog for your use in fine tuning that ratio so you can easily home in on a height that matches your radius and gives a 5000 mm spherical radius.

This produces a shallow spherical segment to use for the bottom section.

That patch was merged into the tank model cutout.

That same patch model was also used as the tank roof.
The vertical stiffeners and hoop members are modeled as offset eccentric beams in order to represent the full stiffening effects of these members.

The construction of the frame and simple supporting structure are typical of many other models covered in much more detail in other samples so are not discussed further. The main objective of the sample model is to illustrate hydrostatic loading.

LOADING

Hydrostatic pressure loads are applied by:

Selecting the tank wall by group including all plate element below the zero load level (surface)
Using Set loads from the pop-up menu and then open the Hydrostatic load tab
Enter 17000 mm for the zero load level (the seam at the roof/wall intersection) for a FULL tank
Direction of increasing normal force is Y negative.
Enter 9.789E-6 N/mm³ for the intensity (fluid weight density).

For a tank less full, say 10000 mm, you would select ONLY those elements below 10000 mm, and set the zero load level to 10000. Alternatively you can set the zero load level to 10000 but select all tank wall elements even above the fluid level as long as you check the option Pos (+). This ensures that only outward pointing pressure is applied which would be below the 10000 mm level.

The model is loaded here only with hydrostatic pressure since treatment of hydrostatic loads is a main objective of this illustration. It would be a simple manner to include dead weight.

From Model/Gross properties: F_Y = 1.173E7 N

Compare this with the weight of water in the tank by volume: 1204.3 M³ x 9789 N/M³ = 1.179E7 N
This compares well with the total tank load from internal hydrostatic pressure.

See the sample file Hydrostatic2D.fem for an example of hydrostatic loads with beams in a 2D model.
DYNAMIC SAMPLE MODELS

AIRPLANE.FEM (FREE AIRPLANE BEAM MODEL)

This is a simulated beam model of an airplane with Gross weight 241,000 pounds and a wind span of ~125 feet.

This is a typical aircraft vibration "stick" model often used for initial investigations in early design stages.

This example illustrates the use of the free vibration solution option to examine naturally unrestrained free vibration.

The stiffness and mass data for this model are more or less arbitrary and chosen provide frequencies and mode shapes typical of an airplane of this type and size.

Notice that the model is restrained on one wing tip only. All models must be initially restrained against free body motion although the restraints will be released (internally) during solution. The restraints not only allow a solution to take place but provide the information that determines the rigid body modes that need to be released to create a fully free solution.

The model could be restrained at any node so long as the restraint does not isolate one part of the model from the other. For this particular model, that leaves mainly the extremities such as wing tip, nose, etc. The only difference is that sometimes convergence is more or less reliable between these selections.

Solve with the "Free vibrations" option selecting to release all six degrees of freedom (full free). Here are the results (Hz) for the 10 lowest modes.

1.616, 1.710, 2.480, 3.338, 4.419, 5.380, 6.633, 7.068, 7.127

The model has 158 vibration modes available (equal to the actual degrees of freedom), however only a few lower are derived here, as is typically the actual interest.

Always choose to derive the minimum modes desired. Models are usually constructed with many degrees of freedom because that improves the accuracy of the modes desired and improves the displayed mode shapes. This, however, makes many more modes theoretically available. Choosing to derive all of those higher frequency modes can take a lot of computer time and the higher frequency modes may not be very accurate or useful. A further issue with too many modes is that the convergence can be a problem with higher modes and no solution at all might be the result. Always choose the minimum number of modes you actually need.

If you have a fast computer you may need to reduce the vibration speed to an observable level. When viewing the modes you can right-click on the mode panel and change the animation speed.

ANTENNASEISMIC.FEM (Seismic analysis)

This is a 120 inch diameter parabolic antenna with focus located at 50 inches from the center. The antenna is to have 12 spokes and 8 rings spaced evenly across the disk.
This sample is intended to demonstrate the complete approach to seismic analysis as well as to illustrate use of certain templates and the surface shaping utility.

GEOMETRY
Focal length: F = 50
From analytical geometry the equation of surface is:
$$R^2 = 4FX$$ or:
$$X = \frac{1}{(4F)}R^2 = 0.005R^2$$

DISH CONSTRUCTION
Use Constructs/QuickModeler with the Cone template.
ZY plane
X, Y, Z = 0, 0, 0
Radius 60 Height 0 (i.e. H=0 makes a flat mesh)
Sections 8 Sectors 12
Beams only
Note: There are other ways to make the initial flat mesh such as making a single 8 segment beam and replicating it 12 additional times in a rotary fashion about its end with all nodes previously selected to make the radial interconnections.

Select all nodes
Use: Utilities/Constructs/Shape surface
Choose the equation for the polar parabola: $$W = W + AR^2$$
Set: 0, 0, 0 as the reference point
Set: W = X
Set: M=2  
Set: A = 0.005

This will make the precise parabolic shape with focus at 50 inches in the ZY plane (standing vertical and pointed horizontal to the right as shown above.

The two 9 inch long support arm attachment nodes are selected then use 'Copy nodes relative' with the option to include connecting beams. Set X = -9.

TOWER

The tower is made using simple standard construction methods. Briefly, the support arm end nodes are selected and 'Copy nodes relative' is used (with connecting beams) to extend 2 posts 100 inches downward to the ground restraint point. The 'Divide' method is used to divide both posts into ten equal parts. Cross beams and diagonals are drawing between the two posts. The complete two post construction is selected and replicated 9 inches farther behind with connections at each node creating the 4 post design. Finally, side cross diagonals are drawn in.

ALTERNATE TOWER

You can make the tower separately in one operation from Utilities/Quick modeler using the 'Box' template with the following settings.

-Corner node 1: 0, 0, 0
-Corner node 2: 9, 100, 15
-Width div=1; Height div = 10; Depth div = 1
-Use Diagonals; Close top

Then rename the group names for the posts and crosses. Save it, reload the dish model and merge the tower into the dish model. You can modify it by adding, joining, or deleting cross members as needed to conform to design.

The antenna, as first constructed, is vertical and pointing horizontal to the right. The pivot is assumed to be where the arms attach to the tower. To change elevation, all the elements in the antenna and support arms are selected along with a node at the pivot point, then use Constructs/Rotate and rotate about the Z axis. Here it is rotated by +30 degrees relative. You can easily re-perform this operation to reset the antenna for other elevations.

The tower is made of steel round tube while the antenna is made of aluminum tube. The focal support beam is polycarbonate material. These properties are set up in the element library.

The model type is set to Dynamic (Edit/Model type) and then is loaded with a 5 pound mass (0.01295035 bugs) attached at the focal point in addition to dead mass of all structure.

SEISMIC ANALYSIS

With 20 modes solved, check Results/Modal data and note that the effective mass in the X direction is 63.88%. Not up to code standards (which typically ask for 90%) but this is a downside of having a very flexible component mounted on a very stiff structure. You would need many more modes (60 would result in 86% and with frequencies up into the audible range!) to reach that level and final results wouldn't be much different. The 20 modes reach high enough relative to seismic spectrums to be sufficient for this model. But it is no more difficult to perform use many more modes if you want. We are using 20 modes.

34.336, 34.608, 35.698, 37.365, 37.755 Hz

Assume the local spectrum is given as:

IBC Sds = 0.900 Sd1 = 0.540 Ts = 0.600

You could easily generate this spectrum using the CADRE Pro shock spectrum generator but this one is already contained in the sample spectrums provided with CADRE Pro.

Use Results/Shock analysis

Set CQC method for combining modes
Set 0.05 for damping
Set the Design Acceleration to 386.09 in/sec\(^2\) (i.e. 1g since our spectrum is constructed relative to 1g)

RESULT SET METHOD
For the first analysis choose 'Result Set' (this combines modes at the internal stress level)

Use the Eye review tool on the dialog to display stresses in individual members.
Choose Normal Stresses
Choose Extreme of either end

The table shows the normal (axial plus bending contribution) stresses in every member in the structure. At the bottom is a summary of the greatest stresses and the member that produces them.

Notice that member S2*13*1 has the largest stress at 6407 psi. This is one of the aluminum antenna segments attached directly to the support arm.

If you performed this same analysis using 60 modes you would get 6415 psi on the same member.
You can use OK to save this result set (*.dtf) so that you can review it directly later using a static version of the model.

LOAD SET METHOD
This method combines the modes at the external load (inertial force) level and produces a set of static loads that can be used to apply to a static version of the model.

Choose 'Load set'
All other settings are the same as above.

Use the review tool on the dialog to see a table of external loads on all mass nodes. At the bottom of the table you can see the total value for \(F_x = 186\) pounds which is essentially the maximum lateral shear in the X direction. The table isn't very useful except as a general overview of inertial forces and quick check of the process.

Use OK to save this load set as a standard static model load set (*.lsb) which can be loaded onto a static version of this model (using Set loads) and solved for displacements, internal loads, and stresses. They will be different stresses since the modes are combined at the external load level rather than the final output level. Many prefer this approach since the load set can easily be combined with dead weight and other external loads for final analysis.

DISPLACEMENT SET METHOD
This method combines the modes at the external displacement level and produces a set of nodal 'specified' displacements that can be used to apply to a static version of the model. Some codes require this type of analysis often with a modified spectrum to assess the relative maximum displacements allowed in the structure.

Choose 'Displacement set'
All other settings are the same as above.

Use the review tool on the dialog to see a table of external displacements on all mass nodes. At the bottom of the table you can see the maximum displacement for \(X = 0.4\) inches at node 89. The table isn't very useful except as a general overview of displacements and a quick check of the process.

Use OK to save this displacement set as a standard static model restraint set of 'specified displacements' (*.bsb) which can be reapplied onto an unloaded static version of this model (using Set bounds) and solved for displacements, internal loads, and stresses. They will be different loads and stresses from the Result set above since the modes are combined at the external displacement level rather than the final output level. However, the main purpose of this assessment would be to check for maximum allowed seismic displacements.
STATIC VERSION

The checks with the dynamic model are only for cursory review. A static model allows many more ways to sort stresses and output extreme values for each element type. You may now have a *.dtf file, a *.lso file, and a *.bsb file for use with a static version of the model where your main analysis takes place.

Typically you would save a separate copy of your dynamic model and convert it to a static version using Edit/Model type. Then, with the static version:

Use File/Results/Review/Shock set to examine internal loads or stresses using the *.dtf file.

Or, use Set loads and load your model with the *.lds file and solve it just like any other static model.

Or, use (unload the model first) Set bounds and reapply the specified displacement set *.bsb and solve.

Once solved, you can use all the results sorting and display methods to present the relevant seismic data.

Examine your design code to determine best how to use each of these results. Typically the code recommends that modes be combined directly at the "parameter of interest".

When you have only one mode, all methods give the same result.

**BLACKBOXDYN.FEM (SHOCK ANALYSIS)**

BLACK BOX SHOCK ANALYSIS

This model is used to illustrate structural analysis of mounted items subject to shock impulses.

The objective is structural analysis of the interior board which could represent a circuit board. There are 4 concentrated masses (Each 1/4 Oz. or 0.01562 lb) on the board as well as the distributed mass of the board itself. The interior board is mounted on 5 aluminum posts. The box is 0.125 aluminum sheet and it is supported on 1/2 in diameter soft mount pads, each of which provides a spring constant of approximately 1000 pound/inch.
FEA MODEL

The model is started in the Quick modeler with the box template. The completed model looks like this with an aluminum box on mounts surrounding a circuit board mounted on posts. You could round the corners and provide other cosmetic features but the development to the level below is sufficient.

The information output is stress in the board, inertial loads on any concentrated masses on the board, and the maximum displacement of the board.

SHOCK SPECTRUM

A typical shock spectrum used for this exercise is contained in the sample spectrums and is called "Sample Log(V) vs. Log(F)". It provides a velocity shock impulse in a log-log scale across a broad spectrum of frequencies.

Solve the model for about 20 vibration modes. Then use Results/Shock analysis.
Open the spectrum as described above.

Then you can create an (1) inertial load set, (2) displacement set, or (3) stress results set. You can view and copy these tables directly or better, "Make" them into saved files for use in more convenient and extensive analyses using a static version of this model.

The complete discussion including Model construction, Shock spectrum construction, and final analysis is contained in the CADRE Pro white paper #005.

**BRIDGE3D.FEM (VIBRATING TRUSS)**

This is a 3D vibration model of a truss structure.

This is used in Exercise 1 to practice viewing a dynamic model.

After solving for 10 modes the following data is displayed from Results/Modal data

**NATURAL MODES**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5.760</td>
<td>0.174</td>
<td>7.82E+01</td>
<td>1.02E+05</td>
<td>0.00</td>
<td>12.73</td>
</tr>
<tr>
<td>7.286</td>
<td>0.137</td>
<td>2.02E+02</td>
<td>4.22E+05</td>
<td>97.20</td>
<td>0.00</td>
</tr>
<tr>
<td>9.292</td>
<td>0.108</td>
<td>1.46E+02</td>
<td>4.98E+05</td>
<td>0.00</td>
<td>79.50</td>
</tr>
<tr>
<td>18.522</td>
<td>0.054</td>
<td>1.63E+02</td>
<td>2.21E+06</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>19.291</td>
<td>0.052</td>
<td>1.05E+02</td>
<td>1.55E+06</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>22.048</td>
<td>0.045</td>
<td>6.61E+01</td>
<td>1.27E+06</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>26.090</td>
<td>0.038</td>
<td>1.21E+02</td>
<td>3.26E+06</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>30.881</td>
<td>0.032</td>
<td>1.58E+02</td>
<td>5.93E+06</td>
<td>0.00</td>
<td>7.04</td>
</tr>
<tr>
<td>35.660</td>
<td>0.028</td>
<td>2.36E+01</td>
<td>1.18E+06</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>36.034</td>
<td>0.028</td>
<td>8.64E+01</td>
<td>4.43E+06</td>
<td>0.00</td>
<td>0.46</td>
</tr>
<tr>
<td>TOTAL</td>
<td></td>
<td></td>
<td></td>
<td>97.20</td>
<td>99.73</td>
</tr>
</tbody>
</table>

This does not represent any actual structure and the properties are more or less arbitrary in order to provide a good model for the viewing exercise.
This model is loaded with the frame dead mass plus an additional mass of 10000 pounds (25.9 bugs) on each of
the 8 nodes P1 through P8.

**CANTBEAM.FEM (VIBRATING BEAM)**

This is the vibrating beam model developed and studied in Exercise 6 of "Getting started" in Help.

The same model is also employed again in Help exercise 7 to examine various levels of partially free and fully
free vibration and it is used again in Help exercise 13 to examine vibration with static loads imposed on the
model.

**NATURAL MODES (Cantilever) - See Help Exercise 6**

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Generalized Mass</th>
<th>Generalized Stiffness</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.085</td>
<td>2.936E-04</td>
<td>1.363E-02</td>
</tr>
<tr>
<td>6.778</td>
<td>2.970E-04</td>
<td>5.387E-01</td>
</tr>
<tr>
<td>18.930</td>
<td>3.029E-04</td>
<td>4.285E+00</td>
</tr>
<tr>
<td>36.997</td>
<td>3.122E-04</td>
<td>1.687E+01</td>
</tr>
<tr>
<td>60.995</td>
<td>3.257E-04</td>
<td>4.784E+01</td>
</tr>
</tbody>
</table>

**NATURAL MODES (Freed by both Y, Zrot DOF) - See Help Exercise 7**

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Generalized Mass</th>
<th>Generalized Stiffness</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.856</td>
<td>2.979E-04</td>
<td>5.530E-01</td>
</tr>
<tr>
<td>18.801</td>
<td>3.047E-04</td>
<td>4.252E+00</td>
</tr>
<tr>
<td>36.669</td>
<td>3.145E-04</td>
<td>4.713E+01</td>
</tr>
<tr>
<td>60.307</td>
<td>3.283E-04</td>
<td>1.101E+02</td>
</tr>
</tbody>
</table>

**NATURAL MODES (Freed by Y) - See Help Exercise 7**

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Generalized Mass</th>
<th>Generalized Stiffness</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.724</td>
<td>2.942E-04</td>
<td>3.452E-02</td>
</tr>
<tr>
<td>9.293</td>
<td>2.983E-04</td>
<td>1.017E+00</td>
</tr>
<tr>
<td>22.899</td>
<td>3.026E-04</td>
<td>6.265E+00</td>
</tr>
<tr>
<td>42.538</td>
<td>3.206E-04</td>
<td>2.290E+01</td>
</tr>
<tr>
<td>68.393</td>
<td>2.978E-04</td>
<td>5.500E+01</td>
</tr>
</tbody>
</table>

**NATURAL MODES (Freed by Zrot DOF) - See Help Exercise 7**

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Generalized Mass</th>
<th>Generalized Stiffness</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.746</td>
<td>2.959E-04</td>
<td>2.631E-01</td>
</tr>
<tr>
<td>15.340</td>
<td>3.014E-04</td>
<td>2.800E+00</td>
</tr>
<tr>
<td>31.930</td>
<td>3.073E-04</td>
<td>1.237E+01</td>
</tr>
<tr>
<td>54.519</td>
<td>3.294E-04</td>
<td>3.865E+01</td>
</tr>
<tr>
<td>83.376</td>
<td>2.937E-04</td>
<td>8.059E+01</td>
</tr>
</tbody>
</table>
NATURAL MODES (Cantilever 30 Newton compression) - See Help Exercise 13

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Generalized Mass</th>
<th>Generalized Stiffness</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.321</td>
<td>2.689E-04</td>
<td>1.097E-03</td>
</tr>
<tr>
<td>6.242</td>
<td>3.110E-04</td>
<td>4.784E-01</td>
</tr>
<tr>
<td>18.489</td>
<td>3.080E-04</td>
<td>4.157E+00</td>
</tr>
<tr>
<td>36.585</td>
<td>3.153E-04</td>
<td>1.666E+01</td>
</tr>
<tr>
<td>60.600</td>
<td>3.279E-04</td>
<td>4.754E+01</td>
</tr>
</tbody>
</table>

NATURAL MODES (Cantilever 30 Newton tension) - See Help Exercise 13

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Generalized Mass</th>
<th>Generalized Stiffness</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.458</td>
<td>3.148E-04</td>
<td>2.643E-02</td>
</tr>
<tr>
<td>7.270</td>
<td>2.900E-04</td>
<td>6.052E-01</td>
</tr>
<tr>
<td>19.360</td>
<td>2.986E-04</td>
<td>4.419E+00</td>
</tr>
<tr>
<td>37.405</td>
<td>3.093E-04</td>
<td>1.709E+01</td>
</tr>
<tr>
<td>61.388</td>
<td>3.236E-04</td>
<td>4.814E+01</td>
</tr>
</tbody>
</table>

ELASTDYN.FEM (GROUND/STRUCTURE INTERACTION)

This sample model illustrates the importance of the ground/structure interaction when dealing with a relatively rigid structure on a mat foundation. The most fundamental modes involve significant motion of the foundation on the soil.

Another 2D sample model groundspring.fem demonstrates ground interaction with isolated spread footings while this sample model represents a 3D structure on a large mat foundation formed with plate elements.

FOUNDATION

W = 300 inches (z direction width for this model)
L = 600 inches (x direction width for this model)
Embedment = 36 inches (depth of foundation below soil surface)

SOIL GEOLOGICAL PROPERTIES:

Soil Poisson ratio = 0.20
Soil weight density, \( w = 0.0553 \) lb per cubic inch
Soil shear wave velocity, \( V = 3607 \) inch per sec
Gravity, \( g = 386.02 \) inch per second squared
Shear modulus, \( G = \left(\frac{w}{g}\right)V^2 = (0.0553)(3607)^2/386.02 = 1863.47 \) pound per square inch

APPLY ELASTIC FOUNDATION BOUNDARY CONDITION

You can use CADRE Pro's built-in foundation calculator or use standard foundation stiffness formulas (e.g. John P. Wolf, "Foundation vibration analysis using simple physical models").

To use the build in foundation stiffness feature:
Select the entire foundation's elements, plates only (e.g. use Select/By group and choose the pre-assigned name "foundation").

Use Edit/bounds from the main menu bar or right-click and use Set bounds

Whenever you request to set boundary restraints and have 'Elements' selected rather than nodes, it is presumed that you want to apply spring stiffness to the nodes of those selected elements. Therefore, you are presented with a spring stiffness distribution dialog.

You can apply springs directly as needed. In the case of foundations there is a foundation stiffness calculator that will develop stiffness values based on the foundation geometry and the soil properties.

Click on the Footing stiffness button on the spring stiffness dialog.

Set Type ="Mat foundation" and check "Embedment"

Set Shape = "Rectangular"

Enter W (Z width) = 300 and L (X length) = 600 and set embedment to 36 inches

Set Poisson = 0.2 and G=1863.47

The foundation calculator coordinate system is always as shown on the diagrams with Y normal and vertical to the foundation, X along the length, and Z along the width. If your model was different you would need to use the "Map axes to" feature to align the results with your model. In our case the model system is the same.

Calculate and then OK to import the relevant items for this type of selection.

These are the gross total values as they would apply to the entire mat foundation, CADRE Pro will parse these total values to the nodes of all selected elements on the foundation.
This places the above translational values (Kx, Ky, Kz) into the fields for X, Y, and Z.

Check X, Y, and Z but NOT the Normal box.
Do NOT check the compensation box.
Do NOT check the projected boxes.
Press Assign and the foundation values are parsed to all the nodes of the foundation.
An alternative approximate method (IF all plate elements were approximately the same size and shape) would be simply to allot the spring total values to the 231 foundation nodes.

LOADS
Live load is 3500 pounds (9.067 mass units) on each of the upper floor node points and none on the foundation floor.
Dead mass of foundation and structure is added to this load to create this mass condition.
Both lateral directions and the vertical direction are included in the degrees of freedom.
The basic structure and its loading are not sized for reality. They were selected more or less arbitrarily in order to get some reasonable loads into the elastic foundation which is the subject of this example.
Concrete reinforcement is not considered.
The model is solved for the first 6 modes (it is a large model and takes a few seconds to calculate).

FORCEDRESPONSE.FEM (FORCED RESPONSE)
This is the simple forced response problem developed and solved in Help "Getting Started" Exercise 27.

In exercise 27 you construct a built-in beam and solve it for displacement and stresses resulting from a rotating mass applied at the center. This is the substantiation approach where you need to prove the structure under a specified forcing function. In Help, exercise 28, the design approach is demonstrated in which you must design a structure and forcing level that will achieve a desired performance.
If you go to **Options/Settings** on the **Model tab** and set the node circle 'Max size factor' to 3.0, the node circles will resized proportional to the amount of mass applied. It isn't necessary but provides a better impression of this dynamic model.

The exercise emphasizes the following:

- Dynamic response to single point periodic forces
- Specified displacement type restraints as input files
- Working between dynamic and static versions of the same model
- Consistent units in the Mpa system

Although this is a very simple model, the techniques are applied in the same way no matter how complex the model and no matter how complex the actual structure.

The beam is 2500 mm long and made of a steel 25 x 25 x 2 mm square section from the Australian sections database.

- Mass of the Flywheel at the center = 20Kg = 0.02 Ton
- The masses at the 2 at mid-sides = 10Kg = 0.01 Ton each
- Beam mass is 1.363 Kg/meter on 2.5 meter = 0.003407 Ton total distributed
- Total mass check with all in: 0.043407 ton
- The Flywheel is 6mm off balance

Force in Newton = \( M \omega^2 \)

\[
F = 0.02 \times 6 \times (2\pi)^2 \times \text{Freq}^2
\]

<table>
<thead>
<tr>
<th>Freq</th>
<th>Force(N)</th>
<th>H(mm)</th>
<th>Stress</th>
<th>RPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>75.80</td>
<td>3.80</td>
<td>38.24</td>
<td>240.00</td>
</tr>
<tr>
<td>5</td>
<td>118.44</td>
<td>10.98</td>
<td>113.29</td>
<td>300.00</td>
</tr>
<tr>
<td>5.897</td>
<td>164.74</td>
<td>44.72</td>
<td>470.73</td>
<td>353.82</td>
</tr>
<tr>
<td>7</td>
<td>232.13</td>
<td>14.85</td>
<td>150.97</td>
<td>420.00</td>
</tr>
<tr>
<td>8</td>
<td>303.19</td>
<td>9.76</td>
<td>94.51</td>
<td>480.00</td>
</tr>
</tbody>
</table>

See exercise 27 in getting started for details of construction and analysis.

**GIMBAL.FEM (GIMBAL INSTRUMENT DESIGN EXERCISE)**

This model represents an instrument that is to be designed to suspend a spherical 0.2Kg mass in all 3 rotary degrees of freedom at a frequency of 1 cps in all rotary directions. The frame is designed from OD 2.5 mm 6061 solid aluminum bar.
The system is modeled in a metric (Mpa) system with Newton and millimeter units. The consistent mass unit for the Mpa system is the ton (1000 Kg).

Generally this would be a trial and error solution trying various levels of structural support size and torsion springs. One can start fairly close to the final solution by estimating the mass of structure and ball inboard of each spring set and doing a rigid body classic solution for each spring set.

Extract section and material properties from Basic Shapes module. Use the Material file for Mpa in mass units.

\[ W/L = 1.331741 \times 10^{-8} \text{ton/mm} \]

Mass and inertia in ton and millimeter units are shown in the following table.

<table>
<thead>
<tr>
<th></th>
<th>OG</th>
<th>IG</th>
<th>Ball</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>4.18E-06</td>
<td>4.83E-06</td>
<td>0.0002</td>
<td>2.09E-04</td>
</tr>
<tr>
<td>Ix</td>
<td>5.23E-03</td>
<td>8.71E-03</td>
<td>0.002</td>
<td>1.59E-02</td>
</tr>
<tr>
<td>Ly</td>
<td>5.23E-03</td>
<td>4.90E-03</td>
<td>0.002</td>
<td>1.21E-02</td>
</tr>
<tr>
<td>Iz</td>
<td>1.05E-02</td>
<td>3.81E-03</td>
<td>0.002</td>
<td>1.63E-02</td>
</tr>
</tbody>
</table>

\[ \omega = 2\pi f = \sqrt{\frac{K}{M}} \]

The values of K (Newton-mm/radian) for 1Hz (assuming independent modes) can be found as the product sprung inertia M, and \((2\pi)^2\).

\[ K = (2^\ast1.0)^2 \times M \]

where K is the rotational stiffness and M is the sprung rotational inertia.

The sprung inertia is the inertia inward from each spring extracted from the above table.

<table>
<thead>
<tr>
<th>Spring</th>
<th>Sprung inertia</th>
<th>K</th>
<th>J</th>
</tr>
</thead>
<tbody>
<tr>
<td>OG</td>
<td>1.59E-02</td>
<td>6.29E-01</td>
<td>6.03749E-05</td>
</tr>
<tr>
<td>IG</td>
<td>6.90E-03</td>
<td>2.73E-01</td>
<td>2.61436E-05</td>
</tr>
<tr>
<td>Ball</td>
<td>2.00E-03</td>
<td>7.90E-02</td>
<td>7.57456E-06</td>
</tr>
</tbody>
</table>

The value of J (torsional stiffness coefficient) to achieve the required K values is found from

\[ J = K^*L/(2G) \]

where

\[ L = 5 \text{ mm} \]

\[ G = 26060 \text{ N/mm}^2 \]

the 2.0 factor is for the two springs on each part.

The gimbal springs are set up as standard beams with OD 2.5 mm just like the gimbals except that their J values are substituted with those from the table above to achieve the torsion spring constant.

The 3 gimbals frequencies are approximately 1 Hz each and the first structural mode is just above 32 Hz.

Warning: This type of model with reduced J is a situation with a very soft member connected to a relatively stiff member which, for any FEA model, can lead to a large loss in numerical significance. It just manages to slip by with this arrangement. Using a stiffer gimbal of 5 mm for this exercise will yield garbage (depending on microprocessor being used).

Always check the "condition number" displayed on the status bar after calculation.

**GROUNDSPRING.FEM**

This demonstrates the analysis of a 2D schematic representation of a multi-storied building with ground structure interaction.

For a similar 3D model on a mat foundation see the sample model *elastdyn.fem*. 
This objective of this example is to study the effect of ground/structure interaction with isolated spread footings.

All units are in ton-meter-second system

**MASS PROPERTIES**

Total weight of building is 290 tons
First 3 floors have 80 tons (40 tons on each side)
Top floor has 50 tons (25 tons on each side)
Consistent mass is \(25/9.81\) or \(2.549\) mass units on the top two nodes and \(40/9.81\) or \(4.079\) mass units on the remaining nodes
Degrees of freedom include both vertical and horizontal for each mass node

**SOIL AND FOOTINGS**

Soil shear modulus and Poisson ratio: \(G = 2500\) t/m\(^2\), Poisson = 0.3
Footings dimensions: (length, width, embedment) = 3m, 3m, 2m

**STRUCTURAL ELEMENT PROPERTIES**

Elastic modulus = 1410000 tons/m\(^2\)
Column element inertia = 0.00213 m\(^4\)
Floor element inertia = 0.01707 m\(^4\)

---

**Solution 1 with all reactions fixed (no soil/structure interaction)**

\[ f = 0.8637 \ (T = 1.158); \ f = 2.591 \ (T = 0.386); \ f = 4.191 \ (T = 0.239) \]

---

**Solution 2 with all footing spring degrees of freedom included**

Using CADRE Pro's built-in foundation calculator for the described footing:

Select nodes 1 and 10 at the ground, right-click, and use **Set bounds**
In the Boundary dialog, press the **Footing stiffness button**.
Choose Spread footing, Rectangular
W=3; L=3; E=2
Set Soil (Poisson = 0.3) and Shear modulus (G=2500) for the soil
The fixed footing axes, by chance happen to align perfectly to this 2D model, so
Map: X>X; Y>Y; Z>Z
This ensures that Kx goes to X, and Ky to Y, and Kz to Z, etc.
Use the ‘Calculate button’, then **OK**.

\[
\begin{align*}
Kx &= 19500 \\
Ky &= 22698 \\
Kz &= 19500 \\
Krx &= 48178 \\
Kry &= 66927 \\
Krz &= 48178
\end{align*}
\]

Press OK to apply the restraint condition then solve for the first 3 modes

\[ f = 0.8347 \ (T = 1.198); \ f = 2.530 \ (T = 0.395); \ f = 4.136 \ (T = 0.242) \]

---

**Solution 3 with only X, Y, and Zrot footing spring degrees of freedom**

Select nodes 1 and 10 again, right-click, and use **Set bounds**
Change Z, Xrot, Yrot to **Fixed** leaving others as spring

\[
\begin{align*}
Kx &= 19500 \\
Ky &= 22698
\end{align*}
\]
Kz = Fixed
Kxrot = Fixed
Kyrot = Fixed
Krz = 48178

Press OK to apply the restraint condition then solve for the first 3 modes

f = 0.8347 (T = 1.198): f = 2.530 (T = 0.395): f = 4.136 (T = 0.242)

Same, so we need consider only the 2D spring effects X, Y, Zrot for the footings for this type of 2D model

Effective Building Period per ASCE 7

\[
T = T_0 \times \left\{1 + \frac{K_b}{K_L} \times \left[1 + \frac{1}{K_L} \times H^2/K_T\right]\right\}^{0.5}
\]

T_0 = natural frequency without interaction (fixed) = 1.158

K_b = 4 \times (3.1416)^2 \times \frac{W/(g \times T_0^2)}{12} = 870

W = effective weight of building (taken as 0.7 times actual weight) = 203 tons

H = effective height of building (taken as 0.7 actual height) = 8.4 meters

K_L = total lateral ground stiffness = 2 \times K_x = 39000

K_T = total rocking stiffness = (1/2 Ky \times B^2) + 2 \times Krz = 1.231E6

Where B is the span between the footings or between nodes 1 and 10

T = 1.158 \times \left\{1 + 0.0223 \times (1 + 2.2355)\right\}^{0.5} = 1.198 or (f = 0.8347)

This compares with T = 1.198 above from the model calculation (solution 2 with ground interaction).

**HELICALPENDULUM.FEM (PENDULUM)**

Helical spring pendulum (bob weight)

![Helical Spring Pendulum Diagram](image)

This example illustrates:

Merging models (the existing HelicalSpring.fem model is merged and scaled into this model)

Using pinned beam segments to articulate a beam off of a fixed node

Using outlines and other non-structural features to enhance a demonstrative model

The helical spring is made originally from a Quick Modeler template. It is finished with tapered ends using the ‘Taper’ utility (Utilities/Constructs/Taper) and saved. That original spring is one of the dynamic sample files entitled HelicalSpring.fem and is covered in detail there. The bob weight outline (strictly cosmetic) is made using the feature for inserting arcs and circles. These materials are made weightless since they are only cosmetic.

The pendulum model is constructed, and then the spring model is **Merged** at the appropriate points (Nodes 9 and 10 along with one of the extreme left or right nodes on the coil as a reference). The spring here happens to be the same length as the original spring model so the automatic scaling during merging is 1 to 1 and all spring values should be the same as determined in the HelicalSpring.fem model file below.
If the merging length was different then the spring would have automatically scaled in length and coil diameter (but not wire diameter) and its spring constant would be different. Here we are using the identical length of 7.75 inches so the calculations shown in the *HelicalSpring.fem* model apply here.

There is a 0.25 pound mass (i.e. 0.25/386.09 = 0.000647 bugs) representing the bob weight located at the end of the 16 inch long weightless pendulum arm.

The pivoting end-segment of the arm is a pinned beam (with free Zrot) on its Origin end. You can test this by selecting that element (from node 1 to 2). Right-click and then use *Show beam pins*.

The very last segment of coil spring at the upper end attaching to node 9 is also pinned at its attachment end. Node 10 at the lower end of the coil spring is a pinned boundary node (xxx000).

All other element segments are standard beams.

The triangular support system and bob weight outlines are for visual effect and are unnecessary to the analysis.

The spring constant of the coil spring is determined at about 1.4664 lb/inch (See the helical spring model below for details.)

The solution frequency is 3.772 Hz.

One can alternatively just use a simple axial spring element and set it to 1.4664 lb/in in place of the coil spring.

For example:

Go to the element editor mode

Use *Select/Elements/By Group* and choose the *Axial spring* group (it is currently just a non-structural dashed line).

Once selected, use the *Properties* button and assign the axial spring property to the selected *Axial spring* element (effectively putting it into play).

Use *Select/Elements/By Group* and choose the helical spring group.

Once selected, use the *Properties* button and assign the *Line* properties to those elements (effectively removing the helical spring from play).

Now solve and you can see about the same results at about 3.772 cps

One can also just use a simple spring restraint at the pivot node and set it to 1.4664 lb/in in place any of the spring elements.

For example:

Go to the element editor mode

Use *Select/Elements/By Group* and choose the *Helical spring* group (and end segment).

Once selected, use the *Properties* button and assign the *Line* properties to the spring (effectively removing it from play).

Use *Select/Elements/By Group* and choose the axial spring group.

Once selected, use the *Properties* button and assign the *Line* to the Axial spring (also removing it from play).

Go to the node editor mode

Select the pivot node 9 and use the *Bound* button

Set the Y direction to a Spring type restraint and set $K = 1.4664$ lb/in in the value field. All else free.
Choose **Assign** to assign the restraint to node 9.

Now solve and you can see about the same results at about 3.772 cps

One can include the distributed mass of the arm. However, If the helical coil is active then first ensure the W/L term for the spring wire segments is zero, otherwise there will be many spring modes to consider. (Unless, of course you want to consider the spring vibration modes)

Apply arm dead weight:
With nothing selected, right-click and use **Set mass**.
Choose the **Dead Mass** tab
Choose for the **Merge with exiting** option.
Enter 386.09 for the **Gravity constant** or choose it from the list
Choose the Y degree of freedom
Click on the **Apply dead mass** button
Click on **OK** to keep the new mass system.
Now solve and you will have several modes of vibration including for arm bending modes.

**HELICALSPRING.FEM (HELICAL SPRING DYNAMICS)**

Determine the linear (longitudinal) vibration modes of the spring.

The spring specification is:

- **L** = Length = 7.75 inches
- **D** = coil diameter = 2.0 inches
- **N** = Number of turns (total) = 15.5
- **d** =Wire diameter = 0.1 inch
- **G** = shear modulus = 1.15E7 lb/in^2 or psi.
- The spring is tapered over the last inch on each end
- **w/L** = 2.222677 E-3 lb/inch (force units)
- **m/L** = 5.75689 E-6 bug/in (mass units)

**INITIAL MODEL**

This spring was modeled in the Quick Modeler with the Helix template

- Spring axis normal to the ZY plane (parallel to the X axis)
- Center end node (0, 0, 0)
- **Radius** = 1.0
- **Length** = 7.75
- **Turns** = 15.5 (2 turns per inch)
- **Segments** = 310 (20 segments per turn)

**TAPER**

After creation, change the extents and grid (**Edit/Extents and Grid**) to a grid width of 0.25 inch to aid in selecting the nodes for the taper operation.
The ends were tapered by frame-selecting the nodes of the last inch of each end of the helix and using Utilities/Constructs/Taper.

The taper axis is parallel to the X axis and is tapered in both lateral directions (i.e. radially). For the axis location, use the center end node setting \((Y= 0, Z= 0)\). The taper ratio is set as zero indicating a taper all the way to the centerline axis.

In this case, the taper ratio is the radius of the last selected node on the right (zero) to the radius of the first selected node on the left.

Using the both option with a zero taper ratio tapers the spring radially from the maximum at the left end of the selection to zero at the right end of the selection.

When tapering the left end, check the Reverse check box to taper toward the left.

The taper causes the end node of the wire to be collapsed down to the center line so that it overlays the existing center node. Select all nodes, or just the end ones, right-click and use Attach nodes to correct the overlap by making the overlapping pair into a single node.

The attachment operation causes the last wire segment on each end to be oriented using one of its own end nodes which cause an 'invalid element' error. Zoom in on each end one at a time, select the end segment and...
change its orientation node to one any other node (just so it isn't collinear with the segment). The following picture shows a node selected as the orientation node for the right hand end segment of the spring.

![Node Selection](image)

**ELEMENT LIBRARY**

THE ELEMENT LIBRARY IS SET UP WITH THE FOLLOWING STEEL WIRE PROPERTY.

<table>
<thead>
<tr>
<th>Description</th>
<th>Type</th>
<th>E</th>
<th>G</th>
<th>W/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid Round D = 0.100</td>
<td>Standard beam</td>
<td>2.9E+07</td>
<td>1.15E+07</td>
<td>0.002222677</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Area</th>
<th>Iy</th>
<th>Iz</th>
<th>J</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.007853982</td>
<td>4.908739E-06</td>
<td>4.908739E-06</td>
<td>9.817478E-06</td>
</tr>
</tbody>
</table>

Select all the segments in the model and assign the library property.

**INITIAL RESTRAINTS**

The two end nodes can be set as pinned reactions but one of them should also have the Xrot degree of freedom fixed as well to prevent any rigid body spinning about the spring axis.

The model is created initially as a STATIC model. It must be changed to a dynamic model, but first use the static version to solve for the spring constant which you will need later for verification purposes.

Change the restraint at the right end to the *Specified Displacement* type and set its value to 1.0. Then solve the model and check the load (reaction) at that end node. The value should be 1.4664 lb so that the spring constant, K is 1.4664 lb/in. Be sure to change the restraint at the right end node back to the simple pin before the next operation.

**DYNAMIC MODEL AND MASS APPLICATION**

The model is changed to a dynamic model from *Edit/Model type*. Use *Set mass* and open the Dead Mass tab. Choose the X degree of freedom only since we are interested in longitudinal modes. The library entry dead weight parameter W/L is in force units so you need a constant of gravity. From the drop down list choose the 389.09 gravity constant value which will convert the w/L value to consistent mass units as it is applied to the model. Use *Apply dead mass*. Read the total mass from the metrics panel as: 4.90760E-04 lb-sec²/in

**CLASSICAL SOLUTION AND COMPARISON**

Calculated (classical) spring constant (spring rate) based on a constant diameter spring end to end is:

\[
K = \frac{d^4G}{32ND^3}
\]

Applying this formula with the maximum N=15.5 gives 1.15 lb/in. If it is assumed that the taper regions are rigid then the effective value of N would be 11.5 which would give 1.5625 lb/in. So our measured value of 1.4664 lb/in appears reasonable and accounts for the taper effects.

So use

K = 1.4664 lb/in for the classical solution formula.

Frequency by classical vibration formula (for an un-tapered spring) is

\[
F = \frac{C}{2\sqrt{\frac{K}{M}}}
\]
Where

\[ C = \text{mode number 1, 2, 3} \]
\[ K = \text{Spring rate} = 1.4644 \text{ lb/in} \]
\[ M = \text{Total mass of the spring (check \textit{Gross properties})} = 4.90760E-04 \text{ lb-sec}^2/\text{in} \]

<table>
<thead>
<tr>
<th>C</th>
<th>Classical</th>
<th>Model</th>
<th>Mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>27.331</td>
<td>28.416</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>54.663</td>
<td>56.440</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>81.994</td>
<td>83.699</td>
<td>7</td>
</tr>
<tr>
<td>4</td>
<td>109.326</td>
<td>109.744</td>
<td>8</td>
</tr>
</tbody>
</table>

**COMMENTS**

The model results presented above are only the pure axial vibration modes, selected by animation. There are other lateral modes dispersed between these modes. The lateral modes tend to come in pairs, one for Y lateral and one for Z lateral at approximately the same frequency.

The classical results are not the best estimate. Although the realistic K value was used in the classical formula, it assumes the total mass is also from a constant diameter spring. The model results, which account for all these issues, should be better. This is the usual case where we use approximate classical methods to help give confidence in the more precise results from the finite element analysis.

**OTHER RESTRAINT OPTIONS**

One can eliminate all the sideward vibration modes by restraining the extreme front and back (+Zmax, -Zmax) nodes on the coil with "Fix Yrot" and by restraining all the top and bottom nodes (+Ymax, -Ymax) with "Fix Zrot". This will tend to suppress the lateral modes allowing one to investigate more easily the longitudinal ones.

**QUARTZBEAM.FEM (VERY SMALL STRUCTURES)**

This is a model of a Quartz crystal vibrator beam modeled with beam elements.

The quartz beam is 4 mm long by 0.4 mm wide by 0.15 mm thick.

This model shows:

- that the size of a structure doesn’t affect and the modeling and solutions methods
- the technique of building a beam that can display the torsional aspects without the additional indicating features affecting the results
- the technique of global loading both mass and rotational inertia

The neutral axis of the beam is the only structural item in the model. The lateral elements are massless (W/L=0) and installed just to enable one to see the torsional aspects of vibration. The longitudinal edge elements are just 'line' elements with no properties and no mass.

**Material properties:**

- Mass density = 2.649E-9 Kg/mm³
- \( E = 7.65E-4 \text{ Mpa} \); \( G = 3.114E4 \text{ Mpa} \)

**Section profile properties (solid rectangle)**

- \( t \) (thickness) = 0.15 mm; \( w \) (width) = 0.4 mm
- \( A \) (area) = 0.06 mm²
- \( I_y = 0.0001125 \text{ mm}^4 \)
Iz = 0.0008 mm^4  
J = 0.000343 mm^4  
Quartz density = 2.65 g/cm^3 = 2.65E-9 Ton/mm^3  
(The parameter, Ton, is the metric ton = 1000 Kg)  
W/L (in mass units) = (2.65E-9 *0.06) = 1.589E-10 Ton/mm

Load mass on this model with the global loading wizards.

Select no elements and no nodes. Use Edit/Mass or right-click and choose 'Set mass'

Open the "Mass tab" select all three degrees of freedom, 
Check Replace existing and use the Apply dead mass button.

The translational mass inertia is now applied.

Immediately open the "Dead inertia tab" and check the Xrot box.

There are two options possible:

Option #1 (manual calculation of the torsional value for inertia per unit length)

The conversion constant for torsional (Xrot) inertia (per unit length) is the value needed to derive rotational inertia from mass about the local X axis of the beam. For a rectangle 0.4 by 0.15 the mass rotational inertia per unit length in Xrot degree of freedom is:  
I_x = (W/L)/(1/g)(a^2 + b^2)/12  or  (Mass/L)(a^2 + b^2)/12

So the constant part is 1/(g(2^2 + 15^2))/12 = 1.520833E-2  (where g=1.0 since our W/L is already in mass units)

Enter this factor (1.520833E-2) in the Xrot constant box.

Option #2 (Use the section properties to calculate mass rotation inertia)

Check the box Use section geometry and choose a gravity constant (1.0 in this case).

This method works most conveniently in the torsional inertia degree of freedom (Xrot) but not so well if one needs the pitching (Zrot) or Yawing (Yrot) rotary degrees.

Check Merge with existing to combine this rotary inertia with the exiting inertia due to the concentrated mass you already applied.

Press the Apply rotary inertia button and then OK to exit the global loading dialog.

Solve Dynamic/Vibrations (retaining 10 modes)

1) 8129 Hz = First flatwise bending  
2) 21678 Hz = First lateral bending  
3) 50800 Hz = Second flatwise bending  
4) 131489 Hz = First torsion  
5) 135468 Hz = Second lateral bending  
6) 14188 Hz = Third flatwise bending  
7) 277298 Hz = Forth flatwise bending  
8) 278357 Hz = Second torsion  
9) 335783 Hz = First axial compression  
10) 378348 Hz = Third lateral bending

Some classical solution comparison:

From Roark 'Formulas for stress and strain' calculate (or see Help "Reference formulas")

8147 Hz = first flatwise bending  
131452 Hz = first torsion  
335783 Hz = first axial compression

It is likely that including the Yrot and Zrot (yawing and pitching) inertias of each segment would improve accuracy a small amount but this is rarely needed since the same improvement can be obtained more readily by just increasing the number of segments.
EXPERIMENTAL RESULTS

When comparing actual test results of cantilever beams with high frequencies one should recognize that it is virtually impossible to infinitely restrain such beams in real situations so that they perform as cantilevers at high frequencies and that can make significant difference in the results. For better comparison with tests at high frequencies, you can try to represent the actual support stiffness using stiff spring boundary nodes.

QUARTZBEAM-P.FEM (VERY SMALL STRUCTURES)

This is a model of a Quartz crystal beam modeled with plate elements.

This is the same as the model Quartzbeam.fem which was a beam idealization.

The beam is 4 mm long by 0.4 mm wide by 0.15 mm thick.

This model demonstrates that

- Although the plate model may appear more realistic than the quartzbeam.fem model, it is actually less accurate and takes much longer to solve.
- When idealizing a structure one should give precedence to accuracy and reliability over realistic appearance!

PROPERTIES

Mass density = 2.649 gram/cm\(^3\)
Mass density = 2.649E-9 Ton/mm\(^3\) in consistent units
E = 7.65E-4 Mpa; G = 3.114E4 Mpa
T (thickness) = 0.15
W/A in mass units = 3.9735E-10 Ton/mm\(^2\)

MASS LOADING

One can load this model with the global loading wizards.

Go to the Nodal Editor mode. Select no nodes. Use the Mass button on the editor panel or just right-click and use Set mass

Open the Mass tab, select all three degrees of freedom, check Replace existing and use the Apply dead mass button. Press OK.

This is one of the rare examples where the Power algorithm is far more convenient than the standard default QR algorithm. There are over 1000 degrees of freedom (and as many modes) but we are only interested in a few of the lower ones. The QR algorithm will spend many minutes deriving all 1000 frequencies, then extracting the mode shapes just for the ones you choose to retain. On the other hand, the Power algorithm will immediately begin extracting both the mode frequency and shape one-by-one working only on the modes desired.

In Options/Settings open the Vibrations tab and choose the Power algorithm. Then exit with OK. Sometimes you have to tinker with the number of iterations and perhaps the tolerance. In this case, the default 1000 maximum iterations and the 10\(^{-8}\) tolerance should be adequate. Greater maximum iterations and larger tolerance improves convergence, and the probability of obtaining a solution, but may sacrifice accuracy.

Solve Dynamic/Vibrations (retaining 10 modes)

1) 8193 Hz = First flatwise bending
2) 22727 Hz = First lateral bending
3) 51316 Hz = Second flatwise bending
4) 135987 Hz = Second lateral bending
5) 143788 Hz = Third flatwise bending
6) 158391 Hz = First torsion
7) 282262 Hz = Forth flatwise bending
8) 336461 Hz = First axial compression
9) 357165 Hz = Third lateral bending
10) 467774 Hz = Fifth flatwise bending

CLASSICAL
From Roark Formulas for stress and strain calculate (or see Help Reference formulas)
8147 Hz = first flatwise bending
131452 Hz = first torsion
335783 Hz = first axial compression

In particular, the torsion and lateral bending modes are difficult to obtain accurately for this type of model without a much greater mesh density. The beam element model (Quartzbeam.fem) however is very precise with very few elements and very quick to solve for many more modes so for that model, the QR algorithm is usually better.

EXPERIMENTAL RESULTS
When comparing actual test results of cantilever beams with high frequencies one should recognize that it is virtually impossible to restrain infinitely such beams in real situations so that they can perform as true cantilevers at high frequencies and that can make significant difference in the results. For better comparison with tests at high frequencies, you can try to represent the actual support stiffness using stiff spring boundary nodes.

RING.FEM (FREE-FREE SYSTEMS)
This model shows the various vibrating modes of a free unrestrained ring.

The ring is 24 inch diameter of 1/4 inch diameter solid steel rod.

CONSTRUCTION
This ring can be constructed directly on the screen using _

File/New
Grid and extents:
-12, -12, 0
12, 12, 0
2
Choose 'Dynamic model'

You are automatically in the Nodal Editor, Isometric view: with the snap tool active:.
Using both Ctrl and Alt keys and left mouse button set points (0, 0, 0); (-12, 0, 0); (0, 12, 0) Select (0, 0, 0) as the center node) then the left, then the top node in order.
With all three nodes selected, Use Utilities/Insert special structures/Arcs and circles
Check: 'Attach nodes'
Enter: 60 in the field for segments
Choose: 'Full circle'
Note: The distance from the first selected node to the second is the radius and start position, the final node orients the circle to the plane in which the 3 nodes lie.

Or, alternatively using the Quick modeler and circles with
Ring template
Set the XY plane
Set the center node to (0, 0, 0)
Set the radius as 12
Set the eccentricity as 1
No. points = 60
Uncheck: spokes (no spokes)

MASS LOADING
The mass was set up by a right-click and choosing Set mass from the pop-up menu.
Or, by using Edit/Mass from the main menu
Then open the Dead mass tab on the global loading dialog.
Set the X and Y degrees of freedom.
Choose a Gravity constant of 386.09 from the list.
And, choose to Replace existing mass on the model.

Note: The conversion constant is necessary because the library W/L parameter for the defined section happens to be in weight (force) per unit length, which was taken from the Mat SampLIF.txt materials file. If it has been set up as mass per unit length (perhaps using the Mat SampLPM.txt file then the gravity factor would be 1.0.

RESTRAINT
For Free vibrations the model still must be restrained but should be set with the least possible restraint to make it stable for solution being careful not to isolate one part of the structure from the other.

The initial restraint for this model is only at a single node at the top most point (Node 3 on this model, or Node 16 if you generated it in the Quick modeler). The single restraint is fixed in all 6 degrees of freedom. This restraint will be virtually released during the Free Vibration solution.

SOLUTION
Solve the model with the Free vibrations method, choosing to free all possible rigid body modes (default).
The sample is solved for 12 modes; notice the repeated pairs of roots caused by the symmetry of the model.

These repeated roots make the modal extraction process numerically very difficult. In fact, by theory, the process should fail to converge but because of the very slight inaccuracies in calculation caused very slight round-off differences, a solution can usually take place.

SCISSORS-DYN.FEM (SCISSOR CONNECTIONS)
This model is constructed in Help, “Getting started”, Exercise 16

It is an example of beams pinned to each other to create a scissor type joint.
The main model is a static one (see the Static samples file scissor.fem)
This is the same file but loaded with dead mass to examine vibration modes about the scissor connection.

**SCREENTABLE.FEM (FORCED RESPONSE DESIGN EXERCISE)**

The purpose of the exercise in Help "Getting started" (exercise 28) is to provide confidence and validation of the periodic forcing dynamic analysis feature in CADRE Pro.

![Diagram of screening table with forces and masses]

It demonstrates the design approach to solving engineering problems as opposed to the substantiation approach. Here, the performance requirement is given and you must develop the input and some of the model requirements. The exercise is adapted from a solved problem in a machine design textbook (Shaum's outline "Theory and problems in machine design").

**PROBLEM STATEMENT**

Part of a processing operation requires a screening table to be reciprocated with amplitude of 0.025 inch at a frequency of 6 Hz. The table is to have 4 leg supports which act as spring beams about their small section dimension. The table is 36 x 36 inch square and is to be vibrated by means of a solenoid vibrator at one end of the table top. The friction in the table is equivalent to a damping factor of 0.05. The table and the working contents are known to weigh 80 pounds.

Assuming the easiest and least force to shake the table would be at resonance, find:

- The length (L) of the legs
- The force (\(F_0\)) amplitude needed to vibrate at the required displacement of 0.025 inch

To begin, choose some arbitrary initial length for the legs and build the model.

The model is effectively rigid except for the spring beam legs and all the mass (including weight of the structure) will be simplistically applied to the table top nodes.

See Help "Getting started" Exercise 28 for details of construction and to review both the CADRE Pro solution and the textbook solution.

**SEISDYN.FEM (SEISMIC SAMPLE)**

This is the dynamic version of the seismic model constructed in exercise 21 of "Getting Started" in Help.

![Diagram of 2D frame with concentrated weights]

This dynamic model is used to develop the quasi-static load case for use in seismic analysis. The static version of this same model is "seisstat.fem". That one will be the used to apply the quasi static load set and solve for the seismic condition.

It is a simple 2D frame of W6x15 steel beams loaded on each rung with a 500 pound concentrated weight (1.295 bugs in consistent mass units).

The model is solved for a few modes (e.g. 10 modes).
Then go to Results/Shock analysis
In the spectrum drop down list, choose the spectrum named:

IBC Sds = 0.900 Sd1 = 0.540 Ts = 0.600

Set CQC method and damping at 0.05.

The above spectrum is per "g" so to get real acceleration units (and consequently real forces) use 386.09 in/sec^2 as the Design acceleration.

One-by-one, check off modes and leave it checked IF it changes the effective mass, otherwise uncheck it.
If the restraints are fully fixed then modes 1, 2, 3, and 4 will be the only effective ones.
If you use the ground spring restraints (see exercise) then modes 1, 2, 3, 4, 6, and 10 will be effective.
See Help, 'Getting started', Exercise 21 for the step-by-step details in developing this seismic solution.

SEISSTAT.FEM (SEISMIC EXAMPLE)
This is the static version of the seismic model constructed in exercise 21 of "Getting Started" in Help. This is made directly from Seisdyn.fem by changing the model type and saving under the new name Seisstat.fem.

This static model is used to apply the seismic displacements or seismic loads and resolve internal loads and stresses.
See Help, 'Getting started', Exercise 21 for the step-by-step details in developing the complete seismic solution.

SPIRALPENDULUM.FEM (SPIRAL PENDULUM)
Spiral spring pendulum

This example illustrates:
- Quick model spiral template
- Merging models
- Using pinned beam segments to articulate a beam off of a fixed node

GENERATE A SPIRAL
The spiral spring was constructed in the Quick Modeler using

Ro = 1.0
Ri = 0.25
Turns = 4.5
Segments = 120

Use Edit/Model type and change it to a 'Dynamic model'
Then save it as a separate model (e.g. spiral.fem).

![Diagram of simple pendulum arm and supports]

The simple pendulum arm and supports were then constructed.

The triangular 'support system' elements are for visual effect and are unnecessary but some node off the beam centerline is needed for orientation and for the upcoming merge.

MERGING MODELS

First select nodes 14, 1, then 2 on the main model with support and beam,

![Diagram of nodes 14, 1, and 2]

Then use Utilities/Merge and choose the spiral model you generated.

Select nodes 8, 122, 1 on the spiral model, Check Keep Aspect ratio and Check Attach nodes. You can leave Full model unchecked since the merging model has no loads or restraints to consider. Then Continue to merge the models.

The points 1 and 2 on the main model and 122 and 1 on the merging model are matched. The other nodes, 8 on the merging model and 14 on the main model serve to the the planes for the models so that they align in the same plane.

MASS

A 0.25 pound mass (0.000647 bugs) is at the end of the weightless pendulum. Select the end of the pendulum, use Set mass and enter 0.000647 in the field for mass and check just the Y degree of freedom. Assign the mass.

THEORETICAL CALCULATION

The theoretical calculated spiral spring torsional stiffness from the mechanical engineers handbook is _

\[ K_T = 8.158 \text{ inch-pounds/radian} \]

Theoretical calculations yield

\[ F = 1.789 \text{ cps} \]

SOLVE

Solve/Vibrations and choose the one and only mode.

The model yields 1.784 cps

If the only interest is the frequency, then a simpler alternative to the spiral spring is a simple pinned beam segment with a spring fixity factor at the pivot end of the segment.
SPRINGMASSSYSTEM.FEM (STUDY IN GENERALIZED MASS)

This is solved for natural frequencies and modes. Afterward it is explained how to apply a periodic vibrating force to one of the mass nodes and determine displacements and internal loads.

\[
\begin{array}{ccc}
\text{m}_1 & \text{m}_2 & \text{m}_3 \\
k_1 & k_2 & k_3 \\
\end{array}
\]

Let these be 1 pound (0.00259067 bugs) concentrated masses separated with springs (K=1 pound/inch) and consider only the X degree of freedom. Consistent mass is 0.00259067 bugs for each concentrated mass (i.e. 1/386).

Even though all the spring properties are the same for this example, three separate entries are made in the element property library just so that three different colors can be shown on the display.

COMPARISON WITH CLASSICAL ANALYSIS (BENCHMARK)

This system is a good one for study and for benchmark validation since it can be calculated precisely by classical methods (i.e. 1.391598; 3.899169; 5.634463)

CASE 1 (Restrained system solution using Solve/Dynamic/Vibration and retain all 3 modes)

Restrained system results from CADRE Pro (Results/Model data)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.392</td>
<td>4.770E-03</td>
<td>3.647E-01</td>
<td>91.408%</td>
</tr>
<tr>
<td>3.899</td>
<td>4.770E-03</td>
<td>2.863E+00</td>
<td>7.488%</td>
</tr>
<tr>
<td>5.634</td>
<td>4.770E-03</td>
<td>5.978E+00</td>
<td>1.104%</td>
</tr>
</tbody>
</table>

And, from Results/Displacements for each displayed mode

Mode 1 Mode 2 Mode 3
1.0 1.0 -0.801938
0.445042 0.445042 1.0
0.801938 -0.801938 -0.445042

CASE 2 (Free system solution using Solve/Dynamic/Free vibration and retain both modes)

Free-Free system results from CADRE Pro

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3.127</td>
<td>5.181E-03</td>
<td>2.000E+00</td>
</tr>
<tr>
<td>5.416</td>
<td>3.886E-03</td>
<td>4.500E+00</td>
</tr>
</tbody>
</table>

For the free-free version of the system, the mode shapes can be determined by reasoning since they must inertially balance, be unique, and be normalized to 1.0. Inertial balancing forces are a function of mass and distance from the mass center. Therefore in order to balance;

Mode 1 Mode 2
+1.0 -0.5
0.0 +1.0
-1.0 -0.5

Using these modes, one can calculate directly the generalized mass for the free-free system from

\[
GM = \{\{\Phi^T\}^T[M]\{\Phi\}
\]

where

[M] is a 3 x 3 square diagonal matrix of masses M1, M2 and M3 with all else zero
(\Phi^T) is the transpose of the matrix of modes listed above. 2 rows 3 columns
(\Phi) post multiplied as columns (3 row x 2 columns)

The result is a 2 x 2 diagonal matrix with the diagonal terms indicating the general mass:

\[
GM1 = 0.00259067 x (2.0) = 0.00518134; GM2 = 0.00259067 x (1.50) = 0.00388601
\]

One could include the trivial rigid body mode if desired and get GM0 = 0.00259067 x (3.0) = 0.00777201
APPLY A PERIODIC FORCE

Apply a 1.0 pound force (X direction) vibrating at f=4Hz on node M3. (i.e. Fx = 1.0*Sin(2*pi*f*t))
Assume the system has an internal damping of 0.05.

1) CREATE THE STATIC MODEL FOR LATER USE

First unload all results File/Results/Remove
Convert the model to a static model Edit/ModelType
and , save under a DIFFERENT name (e.g. SpringMassSystemStatic.fem)

2) SOLVE THE DYNAMIC MODEL FOR MODES

Reload the dynamic model and solve as a fixed vibration system or reload those results.
Go to Results/Forced vibration

3) APPLY THE FORCING FUNCTION

Select node M3 as the point of application of the force
Check all the modes or use the 'Check all modes' checkbox.
Check the X direction and enter 1.0 for the 1 pound force amplitude.
Enter 4 for the frequency in Hz
Enter 0.05 as the damping (or 0,05) depending on the language settings of your computer)

Use the review tool next to the 'Make' button on the dialog to see the following table of displacements
restraints that would be created with these settings.

4) SPECIFIED DISPLACEMENT BOUNDARY SET

Force X=1; Y=0; Z=0
Frequency 4 Hz
Damping 0.05
Action node M1

<table>
<thead>
<tr>
<th>Node</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>Xrot</th>
<th>Yrot</th>
<th>Zrot</th>
</tr>
</thead>
<tbody>
<tr>
<td>R1</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
<td>Fixed</td>
</tr>
<tr>
<td>M1</td>
<td>3.322E+00</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
</tr>
<tr>
<td>M2</td>
<td>1.217E+00</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
</tr>
<tr>
<td>M3</td>
<td>-2.145E+00</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
</tr>
</tbody>
</table>

Select the MAKE to create and save the boundary displacement set.
At the prompt, enter a file name for the displacement set. (e.g. springmasssystem.bsb)
After saving, the confirmation shows the total work supplied to the system by the superposition of the 3 modes,
each at its amplified (or reduced) amplitude to be 8.557 in-lb.
You could solve and save other force and frequency cases and save them at this point.
Use the Close button to exit the dialog.

5) SOLVING FOR THE FORCED RESPONSE RESULTS

Reload the static version of the model you previously saved (SpringMassSystemStatic.fem)
Right-click and use Set bounds from the popup menu.
Use the ‘Get’ button to apply the boundary set you just saved.
Use OK to keep.
Solve as Static/Standard.
First, use **Model/Gross properties** and check the total strain energy in the elements.

Total strain energy in the springs K1, K2, and K3 reads as: 8.557 in-lb is verified.

Use **Results/Reactions** to read the reaction forces on each node.

- M1 = -3.361
- M2 = -2.105
- M3 = 3.322
- R1 = 2.145

Use **Axial loads** to see the forces with each spring.

- K1 = -2.145
- K2 = 1.217
- K3 = 3.322

Similarly you can check displacements, stresses, etc with the model in the maximum displaced state possible from the periodic vibrating force.

No matter what kind of elements, nor how many elements, and no matter how complex the model, the process of checking response of the entire structure to a single point vibrating load is the same and no more difficult.

**EQUIVALENT LOAD SET**

If you want, you can convert the displacement set to an equivalent external load set. With the static model in the solved state use **Utilities/Constructs/Reactions to loads**.

Solve with that set of "Loads" and check that the forces, displacements, and strain energy are identical. For example, 'Axial loads' shows the following

- K1 = -2.145
- K2 = 1.217
- K3 = 3.322

The advantage of converting the saved boundary set to an equivalent external load set is that such sets can easily be combined with other external load cases.

See Exercises 27 and 28 in Help "Getting started" for other examples of forced periodic vibration

**SPRINGMASSSYSTEMALT.FEM (ALTERNATE CONSTRUCTION)**

This is the same structure as sample model "SpringMassSystem.fem" which is used for benchmark evaluation and for studying forced response.

The masses are 1 pound (0.00259067 bugs) concentrated masses separated with helical springs (K=1 pound/inch) and consider only the X degree of freedom. Consistent mass is 0.00259067 bugs for each concentrated mass (i.e. 1/386).

This model has the single element axial springs replaced with helical springs tuned to K=1 pound per inch. The only purpose of such a model would be for presentation finesse since it is much more tedious to construct and requires more resources.

Even though all the spring properties are the same for this example, three separate entries are made in the element property library just so that three different colors can be shown on the display.

For this model, when solving for the free-free modes check only the X degree of freedom restraint in order to release it.

**TORSIONALSHAFT.FEM (TORSIONAL VIBRATION)**

This is a stainless steel shaft with only the distributed torsional inertia of the shaft itself.
This sample illustrates:

- The smart divide feature
- Translation replication
- Rotational replication
- Applying geometric shaft inertia
- Taper method

The lateral beams on the rod are just weightless non-structural rigid parts used so that one can see the angular rotation of the shaft. Otherwise, the mode shape would not be noticeable during animation.

The shaft restrained (fully fixed) at the left end.

The tapered solid shaft diameter increases linearly from $d_1 = 0.25$ OD at the restrained end to $d_2 = 1.0$ OD at the free end.

\[
\text{Determine the first 3 torsional natural frequencies.}
\]

Length = 12.0 inches  
Material: stainless steel 301  
$G = 10.6E6$ (torsional shear modulus)  
$D$ = varies from 0.25 inch to 1.0 inches  
Density = 0.286 lb/in$^3$

**CONSTRUCTION**

Quick modeler - beam template - XY plane

Left point: 0, 0, 0  
Right point 12, 0, 0  
Set only 1 section (the shaft will be divided later AFTER assigning the taper property!)  
Orientation point: 0, 2, 0  

Make it a Dynamic model type.

**LIBRARY SETUP**

Set up Library Basic shape: Round solid shaft 0.25 inch - standard beam  
Set up Library Basic shape: Round solid shaft 1.00 inch - standard beam  

*Add* to one additional element. Name it *Tapered beam and change its type to Tapered beam.*

Select and copy the first item (0.25 OD section) - use $F8$ key  
Select the tapered entry - use $F9$ to paste - choose paste to ‘origin end’  
Select and copy the first item (1.00 OD section) - use $F8$ key  
Select the tapered entry - use $F9$ to paste - choose paste to ‘axis end’
Set some colors for all entries

**Tapered Shaft**
- Sold Round D = 0.25
- Sold Round D = 1.00
- Tapered Shaft

<table>
<thead>
<tr>
<th>E</th>
<th>G</th>
<th>W/L</th>
<th>F/L</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.7E-07</td>
<td>1.06E+07</td>
<td>0.1193314</td>
<td>0</td>
</tr>
<tr>
<td>0.04908739</td>
<td>0.0001917476</td>
<td>0.000001917476</td>
<td>0</td>
</tr>
<tr>
<td>0.7853982</td>
<td>0.04908739</td>
<td>0.04908739</td>
<td>0.00817477</td>
</tr>
</tbody>
</table>

Use **OK** to exit.

**ASSIGN PROPERTIES AND CONFIGURE THE SHAFT**

Open the Element editor mode.

Select the beam model (currently has only one element)

Assign the library property for the *Tapered beam* entry to the entire beam.

Select the beam again.

Right-click and use **Divide**

Divide into 24 sections.

This “smart” divide routine maintains the proper taper from left to right, adding additional entries to the library as needed.

**INSTALL WEIGHTLESS RIGID INDICATORS**

Exit out of any editing mode

Torsional movement about a 1D beam axis is impossible to detect on the screen so non-structural rigid indicators are used so one can see the relative motion of parts of the shaft.

Select all the nodes on the shaft.

Right-click and use **Copy nodes relative**, set Y = 0.125, choose to Add connecting beams and press **OK**.

This creates indicators normal to the shaft pointing upward at every node.

Repeat the above again but this time, use Y = -0.125 to extend the spokes below.

Open the Element Editor mode.

Use **Select/Elements/By type** and choose the *Un-assigned* items.

Assign the 0.25 OD *standard* beam property to these items (indicators). This is arbitrary but should be stiff enough to render the beam simply an indicator and nothing more.

Also you need to make it weightless. This 0.025 OD entry is no longer used for anything else so go to the Element Library and set the W/L property of that entry (0.25 OD, standard beam) to zero to make it weightless.

Assign right end node as orientation for all the indicators, then select the right end indicator and make the left end node as its orientation.

Just to be more representative you can taper the length of the shafts from left to right. Here is how.

Select all the nodes. Right-click and use **Utilities/Constructs/Taper**.
Set up to taper parallel to the X axis by a factor of 4 from left to right starting at point 0, 0.

The indicators should taper upward to 4 times as large on the right end.

Create another set 90 degrees around the shaft by

Select all the indicators **Select/Elements/by Type** and choose the library item that was assigned to them.

Use **Utilities/Clone**. Set all translation values to zero. Set up to rotate 90 degrees on the X axis centered on node number 1. Make one replica. Check the item to attach overlaying nodes.

Looking at the shaft endwise now you see you another set at 90 degrees.
Next, use **Reset** to align the model with the XY plane. Then rotate on the screen X axis with the screen **Xrot** control to precisely 45 degrees. Then use **Utilities/Constructs/Reorient** to redefine permanently the indicators as rotated 45 degrees which may make a better view.

**APPLY TORIONAL INERTIA**

With nothing selected, right-click and_

Use **Set mass**.

Open the Dead inertia tab

Check the option: **Use section geometry**, and

Choose 386.09 as the gravity constant from the drop down list, or just type it in the field.

Select the **Xrot** degree of freedom.

The optional **Use section geometry** feature uses the Iy, Iz, and Area values in the element library to develop a polar radius of gyration for each segment, combines this with the gravity constant, and the local W/L value for the element, then develops a rotary mass moment of inertia.

\[
I_p = mR^2 \\
R^2 = (I_y + I_z)/A = \text{square of the polar radius of gyration} \\
m = (W/L)/G = \text{Mass per unit length} \\
I_p = (W/L)R^2/G = \text{Mass inertia per unit length}
\]

The gravity constant (G = 386.09) is chosen for consistent unit conversion since the W/L value we used in the element library is in force units. If W/L had already been in mass units, then you would select the 1.0 value.

The **Use section geometry** option is mainly for torsional inertia on shafts aligned with the global planes.

Choose **Replace**.

Use the **Apply rotary inertia button**.

The model metrics panel shows \( I_x = 0.000121 \) which is the total shaft polar mass moment of inertia.
OK to exit

Notes:

1) The J value in the element library is the structural torsion stiffness coefficient, NOT the polar area moment of inertia. CADRE Pro calculates the polar area moment of inertia of the cross section as the sum of \( I_Y \) and \( I_Z \). For the round shaft, the result is the same as J but not for most other cross sections.

2) If you had other non-weightless beam elements in the library, you would need first to select all the elements for which you want to apply mass and inertia (shaft) and use Show selected only so that the distribution would apply only to those components.

SOLVE

The first three vibration modes are 824, 10469, 17226 Hz

The animation may be improved by right-clicking on the mode panel and slowing down the animation speed and also by reducing the exaggeration.

Another similar practical sample model is Quartzbeam.fem

**TUNEFORK.FEM (TUNNING FORK MODEL)**

This model was constructed from precise measurement of the dimensions of an actual 440 Hz precision laboratory tuning fork.

There are no W/L terms for the library elements since the global load distribution features were not used.

Each segment was carefully measured along the tines, handle, and yoke. With these measurements, the masses and rotary inertia (Iz in the XY plane) were calculated manually and set for each position.

In order to achieve accurate frequencies, solve with Free Vibrations because at frequencies this high, the real tuning fork is effectively unstrained by a simple hand hold.

Select about 10 modes or so for solution. The first mode is 439.977 or essentially 440 Hz as specified on the instrument itself.

**VIBDISK.FEM (VIBRATING CIRCULAR DIAPHRAGM)**

STEEL DISK DIAPHRAGM
This example illustrates some of the issues related to a perfectly symmetric model which has many repeated roots potentially causing convergence issues.

**GEOMETRY AND PROPERTIES**

Radius (R) = 10 inches  
Thickness (T) = 0.125 inches

![Radius and Thickness](image)

Material: Stainless steel (AISI 410)  
Elastic modulus (E) = 29E6  
Poisson (V) = 0.32  
Area density (W/A) = 0.03525 lb/in²

There is a custom viewpoint (Ctrl_U) set up for this model at Xrot = -60 and Yrot = +35 (in that order) which provides a good perspective of the animated vibration mode shapes. The two-tone coloring is just to facilitate the visualization of the mode shapes during animation.

**CONSTRUCTION**

The best way to make a simple circular disk is with the Quick Modeler using the Spheric option.  
This one is made with the following Quick Modeler settings.

![Quick modeler settings](image)

- **Spheric** template  
- **XY plane**  
- **Plates only**  
- **Center node 0, 0, 0**  
- **Radius = 10; Height = 0.0**  
- **Rings = 8, Sectors = 8, No. iso-adjusted = 2**

The 48 rim nodes on the template are automatically provided a group name "EDGE" by the template. For each boundary condition described, you can select using Select/Nodes/By Group and choose "EDGE" to select and reset the rim boundary condition easily.
CLASSICAL ANALYSIS (ROARK, FORMULAS FOR STRESS AND STRAIN)

\[ F_n = \frac{K_n}{2\pi} \sqrt{\frac{Dg}{wR^4}} \]

Where \( D = \frac{E^*T^3}{12(1-V^2)} \), and \( w = \frac{W}{A} = 0.03525 \)

\( g = 386.09 \text{ in/sec}^2 \) for the inch pound force system.

Roark gives constants \( K_n \) for selected modes shown in the tables below. However, these \( K_n \) values depend on the Poisson ratio which are given in Roark at slightly different values. Roark uses a Poisson ratio of 0.3 for all but the free-free option for which 0.33 is used. Neither values match exactly the 0.32 for our model but are close enough to validate our results within reason.

ELEMENT PROPERTIES

The model is loaded using the global loading dialog with the plate properties in the Element Library. You can set up a Kirchhoff plate property with 0.125 thickness, then use the Material item to import the material properties for the 410 stainless steel material into the element library using the file MatSampIPF.txt.

Select and assign the library property to all the plate elements.

Two properties are listed with the sample model just to provide for two different colors to help visualize the animated modes.

MASS DISTRIBUTION

Choose: Edit/Loads (or Set loads from a pop-up menu)

Open the Dead mass tab; NO compensation; \( G=386.09 \) (choose from list); Degree of freedom = Z (only). Using a \( G \) value other than 1.0 implies that the density (and \( W/L \)) values in the element library were from the file MatSampIPF.txt and so are in force units. If you had imported the properties from MatSampIPM.txt then you would simply use \( G=1.0 \).

Note: This plate mesh configuration has a consistent number of plates connected to each inner node so the standard mass distribution method (no compensation) is adequate. The compensation option often creates smoother distribution for certain poorer meshes where there is a wide variation in connections to each inner node.

SIMPLY SUPPORTED

The edge nodes have a Group Name EDGE which can be used to select them easily for changing boundary conditions.

Use Select/Nodes/By Group from the menu bar to select by group and select the EDGE.

Fix X, Y, and Z leaving Xrot, Yrot, and Zrot free (or just use the Pin button).
Roark gives $K_n$ for each mode $n$ with a simply supported edge as follows:

$(K_n$ is base on 0.30 while we used 0.32 in the model)

<table>
<thead>
<tr>
<th>Mode</th>
<th>$K_n$</th>
<th>$F_n$</th>
<th>CADRE Pro</th>
<th>NODAL LINES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.99</td>
<td>60.27</td>
<td>59.77</td>
<td>Fundamental</td>
</tr>
<tr>
<td>2</td>
<td>13.9</td>
<td>167.89</td>
<td>167.76</td>
<td>One diametric</td>
</tr>
<tr>
<td>3</td>
<td>25.7</td>
<td>310.42</td>
<td>307.72</td>
<td>Two diametric</td>
</tr>
<tr>
<td>4</td>
<td>29.8</td>
<td>359.94</td>
<td>357.00</td>
<td>One circular</td>
</tr>
</tbody>
</table>

Notice that there are some repeated roots due to symmetry.

Repeated roots can cause problems in convergence and in the ability to develop mode shapes and generalized mass results. In this model, as constructed there are two roots at 167.760 cps and they could produce precisely the same mode shape which is not correct. Mode shapes should be orthogonal but it is not always possible to derive them with perfectly symmetric models.

**CLAMPED EDGE**

The edge nodes have a Group Name EDGE which can be used to easily select them for changing boundary conditions.

Use **Select/Nodes/By Group** from the menu bar to select by group and select the EDGE.

Fix all 6 degrees of restraint for the edge nodes (or use 'Fix all').

Roark gives $K_n$ for each mode $n$ with fixed boundary as follows:

$(K_n$ is base on 0.30 while we used 0.32 in the model)

<table>
<thead>
<tr>
<th>Mode</th>
<th>$K_n$</th>
<th>$F_n$</th>
<th>CADRE Pro</th>
<th>NODAL LINES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10.2</td>
<td>123.20</td>
<td>122.95</td>
<td>Fundamental</td>
</tr>
<tr>
<td>2</td>
<td>21.3</td>
<td>257.27</td>
<td>256.17</td>
<td>One diametric</td>
</tr>
<tr>
<td>3</td>
<td>34.9</td>
<td>421.54</td>
<td>417.95</td>
<td>Two diametric</td>
</tr>
<tr>
<td>4</td>
<td>39.8</td>
<td>480.73</td>
<td>476.92</td>
<td>One circular</td>
</tr>
</tbody>
</table>

There are some repeated roots due to symmetry

**FREE EDGE**

Free-free symmetric objects have many repeated roots with identical mode shapes and frequencies, which makes reaching a reliable solution more difficult. Convergence is often not complete so the repeated roots may differ by a small amount. The model must start as restrained to be stable then is freed internally during solution. It is best to restrain it as little as possible, not isolate any part of the model by restraint, and also try to be UN-symmetric by restraint, if possible.

Un-restrain all nodes. Then, at the EXTREME LEFT NODE at the edge, FIX it in all degrees.

Check the option to use the nodal centroid as the rotation center.

Then solve as **Free Vibration** solution option and retain about 20 modes.

Roark gives $K_n$ for each mode $n$ is as follows:

$(K_n$ is base on 0.33 (for the free case) while we used 0.32)

<table>
<thead>
<tr>
<th>Mode</th>
<th>$K_n$</th>
<th>$F_n$</th>
<th>CADRE Pro</th>
<th>CADRE Pro</th>
<th>NODAL LINES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.25</td>
<td>63.41</td>
<td>63.47</td>
<td>63.48</td>
<td>Two diametric lines</td>
</tr>
<tr>
<td>2</td>
<td>9.08</td>
<td>109.67</td>
<td>107.50</td>
<td>One circular</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>12.2</td>
<td>147.36</td>
<td>146.72</td>
<td>146.77</td>
<td>Three diametric lines</td>
</tr>
<tr>
<td>4</td>
<td>20.5</td>
<td>247.61</td>
<td>241.86</td>
<td>241.98</td>
<td>One diametric, one circular</td>
</tr>
</tbody>
</table>

These perfectly symmetric models with repeated roots are very sensitive to inaccuracies due to lack of convergence, especially for the free-free situation. For free-free conditions, try different off center restraints, different numbers of modes, and also try adjusting the convergence criterion in Options/Settings on the vibration tab.
Another option for highly symmetrical free-free vibration analysis is to use the simply supported model as described above and set the Z degree of the rim node restraint as a spring with a very low value (e.g. 0.000001 lb/in) and solve it as a fixed vibration model.

**VIBPLATE.FEM (VIBRATING RECTANGULAR PLATE)**

This model is a simple rectangular aluminum plate 20 inches wide by 10 inches high and 0.1 inches thick. It is simply supported at the edges and vibrating under its own mass. This is the model for exercise 9 of Getting Started in Help.

The inch-pound-second system was used.

**ELEMENT PROPERTIES**

The plate is all the same material although two library definitions were used just to create two different colors to aid the viewing of the modes more clearly. The library element dead weight parameter set for \( W/A \) is 0.0098 pounds (force) per unit area.

**MASS LOADING PROCEDURE**

Go to the Nodal Editor, select no nodes and use the *Mass* button OR just use *Edit/Mass* OR just use *Set mass* from the pop-up menu. Choose the Dead Mass tab and set the Z degree of freedom. The gravity conversion constant to consistent mass unit was 386.09 inches per second squared (If the \( W/L \) parameter in the Library was actually in units of mass per unit length then the gravity constant would be 1.0.)

The ‘Use compensation’ option is to smooth out the load distribution for surfaces where some nodes (not counting edge nodes) have considerably more connections that others. For most structures, the standard distribution without compensation is best, but this plate (plate 3 type mesh) can be improved by compensation since some nodes inner nodes are contacted to 4 plates, others to 8.

**RESULTS (HZ)**

<table>
<thead>
<tr>
<th>without compensation</th>
<th>with compensation</th>
<th>theoretical (Roark formulas)</th>
</tr>
</thead>
<tbody>
<tr>
<td>120.751</td>
<td>120.877</td>
<td>120.8494</td>
</tr>
<tr>
<td>193.143</td>
<td>193.409</td>
<td>193.3500</td>
</tr>
<tr>
<td>313.600</td>
<td>314.199</td>
<td>314.2084</td>
</tr>
<tr>
<td>409.543</td>
<td>411.011</td>
<td>410.8879</td>
</tr>
<tr>
<td>481.815</td>
<td>483.074</td>
<td>483.3976</td>
</tr>
<tr>
<td>481.817</td>
<td>483.692</td>
<td>483.3976</td>
</tr>
<tr>
<td>602.058</td>
<td>604.699</td>
<td>604.2469</td>
</tr>
<tr>
<td>697.356</td>
<td>699.743</td>
<td>700.9265</td>
</tr>
</tbody>
</table>
VIBPRELOAD.FEM (PRELOAD AND THERMAL EFFECTS ON VIBRATION)

The structure consists of an aluminum rod restrained between two vertical steel columns.

This sample is set up in the Mpa force system (i.e. Megapascal, Newton-mm).

OBJECTIVE

The rod is to be checked for vibration in the vertical direction at room temperature, then again after being heated to 100 degrees C above room temperature.

It is assumed that the elastic modulus and thermal coefficient are not significantly affected by temperature within the range considered.

MATERIAL PROPERTIES

The columns are 200 mm tall with 65 x 35 x 3 RHS (Australian steel hollow sections). Use the Australian steel database.

![Section database dialog](image)
The horizontal rod is 250 mm long with 6061 aluminum 10 mm solid round section. Create this in the Basic shapes module and import the 6061 aluminum material from the \textit{MatSampMpaF.txt} material file.

**Dead Mass**

The rod is set up in the Element Library with weight per unit length in force units (0.002089159 Newton/mm), so a conversion factor is needed to convert Newton/mm to consistent units of mass. This factor is 9806.7 mm/sec\(^2\).

The actual 'mass' per unit length for the horizontal bar would be:

\[
m/L = (W/L)/g = 0.002089159/9806.7 = 2.130338E-7 \text{ N-sec}^2/\text{mm per mm}
\]

This property from the element library won't be used directly since we want to apply the mass to the bar alone for this particular analysis.

Select the horizontal bar only.

Then right-click and use \textit{Set mass} or \textit{Edit mass} from the main menu bar. This brings up the global loading dialog. Open the Dead mass tab.

In this mode, with elements selected, you have an intensity field where the mass per unit length is to be manually entered.

Check the "Y" degree of freedom and \textit{Replace existing}.

The mass application command will divide the intensity by a chosen gravity conversion factor, so you have two options:

Choose 9806.7 from the drop down list and enter 0.002089159 in the intensity field, or

Choose 1.0 from the drop down list and enter 2.130338E-7 in the intensity field.

Use the \textit{Apply dead mass} button.
The resulting total mass on the rod alone should be $5.3258 \times 10^{-5}$ N·sec$^2$/mm.

Use **OK** to leave the dialog and keep the mass on the rod.

**THERMAL LOAD**

To use static type internal forces in the model you must enable static loads for vibration analysis. Do this in **Options/Defaults** with the **Vibration tab**.

Check the item "Query to allow the application of static loads and temperature during a vibration solution".

This allows the **Preload** button to appear on the element editor panel for dynamic models just like does for static models. It also provides for a query dialog during solution for the user to specify, apply, or skip static loads and/or temperature.

While in the Element Editor, select the rod elements only and use the **Preload** button on the Editor panel.

Set the preload force to 0.
Set the temperature to 100.

The thermal coefficient = 2.358E-05 per degree centigrade (6061 aluminum). Set the temperature coefficient to 2.358E-5 per deg C.

Alternatively, to set the thermal coefficient, you can import it from the materials file (MatSampMPaF.txt or MatSampMPaF.txt which are in Celsius) or just enter the value directly in the field.

After applying the temperature, right-click and use Show preload from the pop-up menu to see the temperature directly on the screen above each segment of the member to which it is assigned.

Similarly, use Hide preload to hide the temperature labels.

SOLUTION OPTIONS

Use Solve/Dynamic/Vibrations and choose to retain about 10 modes.

At the query choose Skip for the first run and this will ignore all static loading including the internal thermal loading.

Read 718 Hz for the first mode.

Run again. This time, when the static loading dialog appears, check the option to “Include existing preload or temperature” and use the OK button.

Read 497 Hz for the first mode.
This is the effect on frequency caused by the reduction in elastic stiffness resulting from the internal compressive load arising from the temperature change.

If one end of the rod were not restrained by the vertical post there would be no affect, as the temperature could not build up a compressive internal load.

Try changing the temperature to -100 C. The fundamental frequency should increase to 882 Hz.

**XYLOPHONE.FEM (MUSICAL INSTRUMENT)**

Xylophones used in music classrooms are about 1 ½ octaves and usually from Middle C to A in the next higher octave and are made of wood and other materials.

This abbreviated example is from middle C to D in the next octave and made with brass bars. The brass tone bars (20 mm x 2.5 mm) are designed with a specific length to yield the fundamental free-free bending frequencies for each note.

The term "Node" in the following table refers to the dead spot on the bar when it is vibrating in its fundamental free-free mode. Not to be confused with the term 'node' typically used in finite element models!

The values for the node location are the distance from the center of the bar to the 'Node' or dead spot. One can support the tone bars at these node points with minimal interference in the fundamental frequency. These node points are on the bars at the intersection of the dashed line. Mechanics gives 0.276L as the distance out from the center of the bar to the dead spot for any free-free vibrating beam. See Help for basic vibrating beam formulas.

<table>
<thead>
<tr>
<th>Note</th>
<th>Freq</th>
<th>L</th>
<th>L/2</th>
<th>Node</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>264</td>
<td>187.42</td>
<td>93.71</td>
<td>51.73</td>
</tr>
<tr>
<td>D</td>
<td>297</td>
<td>176.70</td>
<td>88.35</td>
<td>48.77</td>
</tr>
<tr>
<td>E</td>
<td>330</td>
<td>167.63</td>
<td>83.81</td>
<td>46.27</td>
</tr>
<tr>
<td>F</td>
<td>352</td>
<td>162.31</td>
<td>81.15</td>
<td>44.80</td>
</tr>
<tr>
<td>G</td>
<td>396</td>
<td>153.02</td>
<td>76.51</td>
<td>42.23</td>
</tr>
<tr>
<td>A</td>
<td>440</td>
<td>145.17</td>
<td>72.59</td>
<td>40.07</td>
</tr>
<tr>
<td>B</td>
<td>495</td>
<td>136.87</td>
<td>68.43</td>
<td>37.78</td>
</tr>
<tr>
<td>C'</td>
<td>528</td>
<td>132.52</td>
<td>66.26</td>
<td>36.58</td>
</tr>
<tr>
<td>D'</td>
<td>594</td>
<td>124.94</td>
<td>62.47</td>
<td>34.48</td>
</tr>
</tbody>
</table>

Solve the model for 10 modes. This yields approximately the frequencies in the table for each bar plus one additional mode since the first "C" bar has a second unsymmetric mode that is lower than the final D' note. Since the center of the C bar is dead in that extra mode it would be unlikely to be excited. When one extends the scale to additional octaves (performance xylophones are 2 and a half octaves or more) there can be more of these unwanted extra vibration modes. Designing a support system to favor the fundamentals and damp or suppress undesired modes is the essence of the design problem.

Since the bars are supported at their 'node' points (dead spots) this model can be solved most effectively as a "fixed" vibration problem. It is also solvable with the free vibration method with approximately the same results but that would defeat the purpose of developing a proper support system since the supports are removed from the system in that method.

The nodes have been renamed with Note definitions and the center node name is set to reflect the frequency expected for that bar.