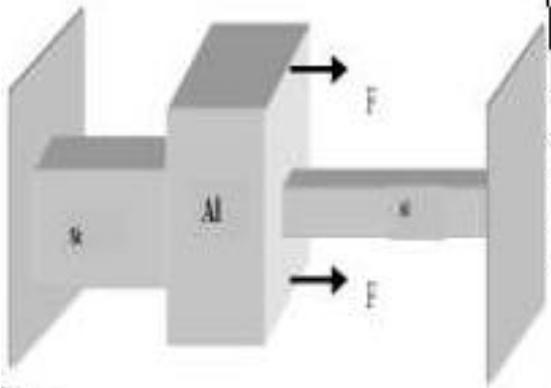
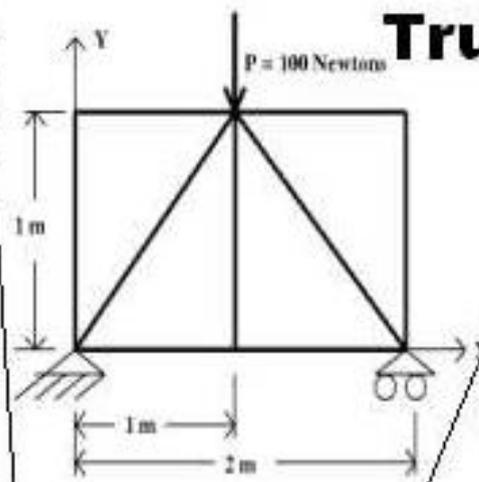


BASIC TUTORIALS IN FINITE ELEMENT ANALYSIS USING MSC/ PATRAN AND MSC/NASTRAN

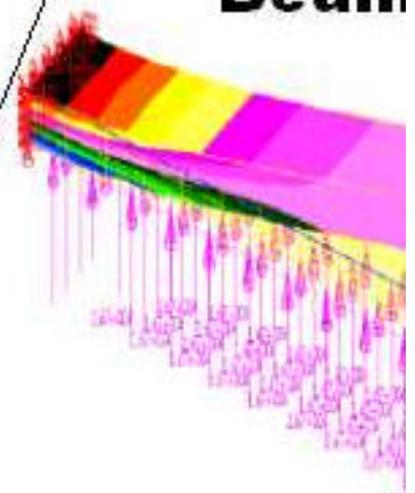
Bars



Truss



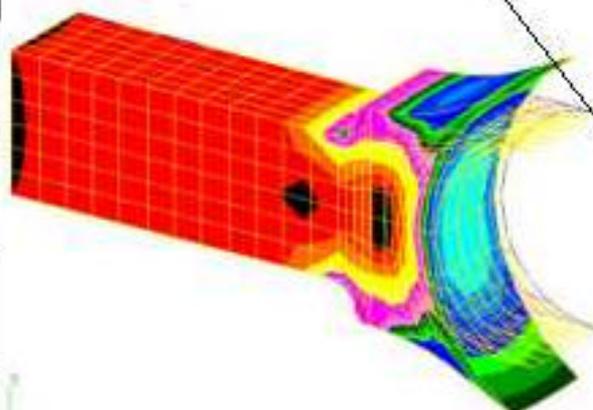
Beam



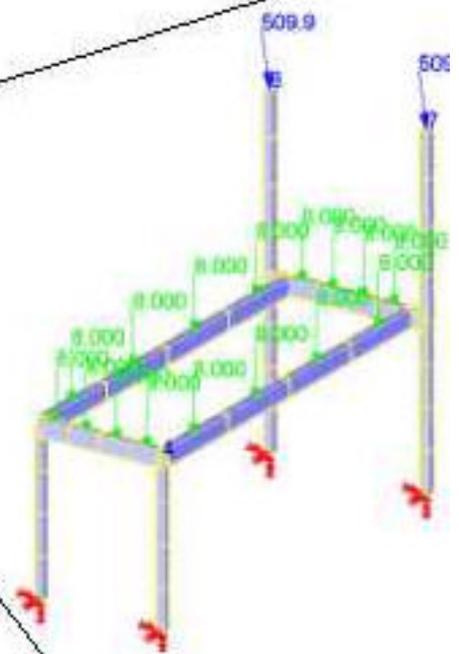
Heat Transfer



Solid



Frame



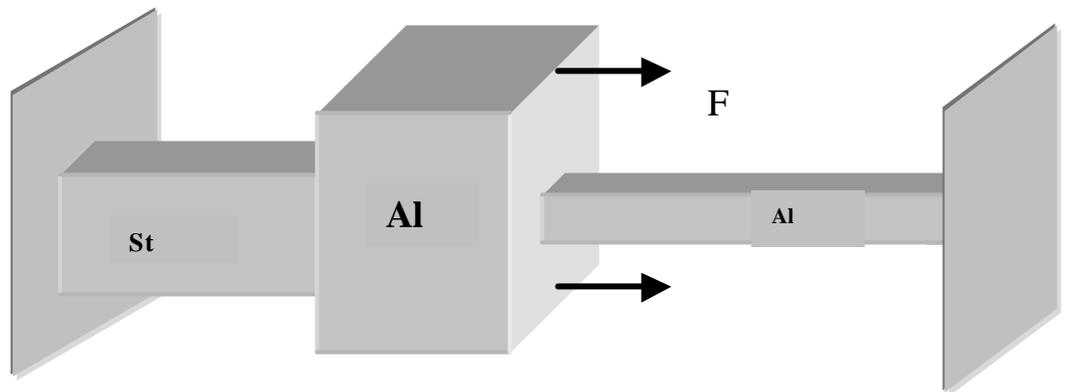
Vibrations



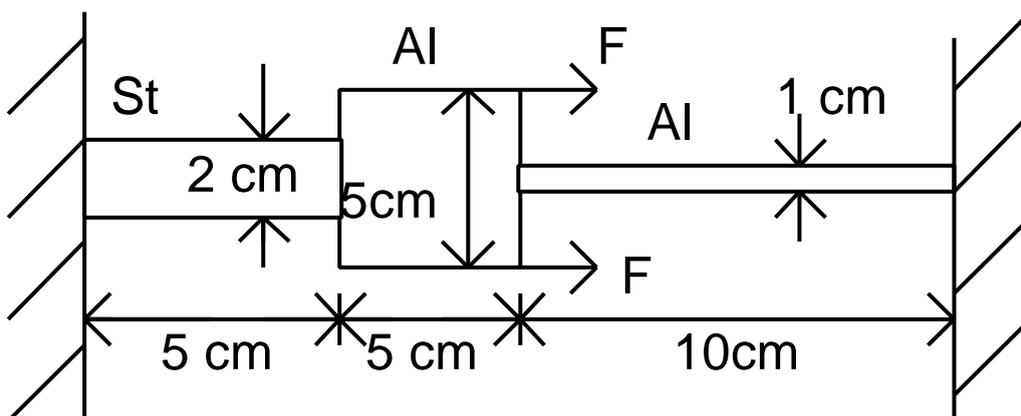
MSC/PATRAN TUTORIAL # 1
MODELING A BAR PROBLEM

I. THE PHYSICAL PROBLEM

In the simple bar problem below, there are three separate sections of the bar. Each section has different properties. The following properties apply, Al → Aluminum, St → Steel, E for Steel = $200 E9$ Pa, E for Al = $70 E9$ Pa. All Bars have square cross section and the right and left ends of the bar are built in. The force "F" = 9000 Newtons



The 2-d model of the problem is shown below.



II. THINKING ABOUT THE MECHANICS

The analytic solution for stresses and displacements for this problem is readily available. Any Mechanics of Materials text will provide equations for the displacements and stresses throughout the bar. The problem is indeterminate because there are two reactions (one at each wall) and only one relevant equilibrium equation ($\sum F_x = 0$). Therefore, it is necessary to use the Mechanics of materials (stress and or displacement) equations as well as the force equilibrium equations to solve the problem.

The normal stress due to axial loading is given by :

$s_{xx} = P/A$, where P is the internal force in the axial direction and A is the cross sectional area of the bar. The displacements are computed from $u = \frac{PL}{AE}$ here L is the bar's length and E is the Elastic (Young's) modulus.

Some basic questions to consider before creating the computational model are:

1. Where will the stresses be tensile and where will they be compressive?
2. What will be the magnitude and direction of the reaction forces?
3. Where will the displacements be greatest?
4. How do the displacements vary along the length (linear, quadratic etc.)?
5. What will the local effect of the concentrated load be on the stresses?
6. Is the model fully constrained from rigid body rotations and displacements?

Answering these questions qualitatively, along with the quantitative analytical solutions for the stresses and displacements, will provide reinforcement that your computational model is correctly constructed.

III. GEOMETRIC AND FINITE ELEMENT MODEL

Some general notes on PATRAN:

A general finite element analysis can be broken down into 3 principle tasks; preprocessing, analysis and post processing. The preprocessing task includes building the geometric model, building the finite element model, giving these elements the correct properties, setting the boundary conditions and loading conditions and finally, assembling these elements into a connected structure for analysis. The analysis stage simply solves for the unknown degrees of freedom, as well as reactions and stresses. In the postprocessing stage, the results are evaluated and displayed. The accuracy of these results is postulated during this postprocessing task.

The Patran and Nastran software together perform all 3 of the principle tasks of a finite element analysis. The pre and post processors are unique to PATRAN itself. However, this package allows the user to do the actual solution analysis on a variety of different packages. At many sites you have the option of using the MSC/Nastran package, which is probably the most widely used solver in industry. Many of the other packages commonly used in industrial settings (ABAQUAS, ANSYS, MARC) are also compatible with PATRAN.

IV. FINITE ELEMENT THEORY

The exact details of the formulation of the rod elements in MSC/Nastran is given in the MSC/Nastran manuals and is somewhat lengthy. However, the basic formulation of an isoparametric 2 node rod element is not difficult and will provide us with sufficient background information to begin to understand the convergence and other accuracy studies. This basic form can be found in any standard text of finite element analysis. For Example see Finite Element Modeling for Stress Analysis, by R.D. Cook, John Wiley & Sons, 1995.

V. STEP BY STEP INSTRUCTIONS FOR MODELING THE BAR PROBLEM USING MSC/PATRAN

Unless you have used the PATRAN software numerous times in the past, the steps shown below should be followed exactly. However, in order to prepare you to do independent finite element work using PATRAN in the future, you are encouraged to go back after you have completed the assignment and investigate modeling options using different PATRAN selections. Also, I encourage you to take notes as you go through this exercise in order to prepare for the time when you will be asked "build a certain geometric structure" or "apply a certain type of boundary condition" with out being given the specific steps for carrying out this task.

The MSC/Patran program is menu driven much in the same way that most Windows programs are driven. Selecting a category from a menu may result in a pull down set of options or in a subordinate menu. Selections in menus may be in the form of buttons to turn on or off, or in the form of boxes which require text. Text entered into boxes may be changed by positioning the cursor at the point of text insertion and either typing the new text or erasing the incorrect text. A standard finite element analysis normally proceeds across the top menus starting with Geometry and ending with Results. Selecting one of these top menus results in a set of menus which allow you to complete that task in the analysis process. Generally, it is best to attempt to proceed from the top of these menus toward the bottom, answering questions as you go.

Preliminaries for using MSC Patran and Nastran normally include:

- 1) Log in to the machine.
- 2) Change to the directory that you wish to contain your results.
- 3) To start the program MSC/Patran, click on Start/Programs/MSC(common) and choose MSC Patran 90.

In the instructions below, the following abbreviations and terms will be used:

TM = Top Menu. This refers to the horizontal menu options residing at the top of the screen after PATRAN has been initiated.

RM = Right Menu. This refers to the menus that pop up after an option has been chosen from the top menu. These menus reside on the far right side of the PATRAN desktop.

SM = Subordinate Menu. This refers to the menus that pop up from options selected in the right menu.

Click = Unless otherwise stated, this indicates a click with the left mouse button.

Boldface will indicate text that occurs in the PATRAN menus.

Italics text will indicate text that you must enter into text boxes in the PATRAN menus or text that you choose in a menu scroll box.

Our first step is to create a new database:

From the TM choose **File**

In the resulting pull down menu choose **New**

A SM called **New Database** pops up

Turn off (no check) **Modify Preferences**

If the new database form has come up showing a directory on a remote computer (as opposed to a directory on the local machine), then switch the directory to the local directory c:\MSC

Under **New Database Name** enter *bar.db*

Click **OK**

The geometry of the structure will be determined next:

From the TM choose **Geometry**

A RM called **Geometry** will result

Set **Action** = *Create*

Object = *Point*

Method = *XYZ*

Set the **Point ID** list to *1*

Set **Reference Coordinate Frame** to *Coord 0*

Turn off the **Auto Execute** button

Enter the following into the point coordinates list:

[0,0,0] [.05,0,0] [.10,0,0] [.20,0,0]

(note that PATRAN will accept either commas or blanks as separators between coordinates)

Click **Apply**

(At this point 4 points should appear on your "bar.db - default_viewport - default_group - entity" main viewport)

The next job is to connect these points to form 3 lines:

While still in the **Geometry** RM,

Set **Action** = *Create*

Object = *Curve*

Method = *Point*

Turn off the **Auto Execute** button if it is on

(for the following, it is assumed that you have created points 1,2,3,4
numbered from left to right in the main viewport. If the numbers are not
in that order, follow the procedure below from left to right regardless of
point numbers)

Click in the **Starting Point List** box
Click on node 1 in the main viewport.
Click in the **Ending Point List** box
Click on the point 2 in the main viewport
Click on **Apply**
(A line will be drawn from point 1 to point 2. This line should be named
line 1)

Click in the **Starting Point List** box
Click on point 2 in the main viewport.
Click in the **Ending Point List** box
Click on the point 3 in the main viewport
Click on **Apply**
(A line will be drawn from point 2 to point 3. This line should be named
line 2)

Click in the **Starting Point List** box
Click on node 3 in the main viewport
Click in the **Ending Point List** box
Click on the point 4 in the main viewport
Click on **Apply**
(A line will be drawn from point 3 to point 4. This line should be named
line 3)

The finite element mesh is specified next:

From the TM choose **Elements**

A RM appears called **Finite Elements**

Set **Action** = *Create*

Object = *Mesh Seed*

Type = *Uniform*

Select **Number of Elements** (button down)

Number = 1

Turn off the **Auto Execute** (button up)

Click in **Curves List** box

Click on the left most curve in the main viewport

(The words "Curve 1" will be added to the **Curve List**)

Click **Apply**

(circles which represent finite element nodes will appear on ends of the curve)

Click **Curve List** box

Click on the center curve in the main viewport

(the words "Curve 2" will be added to the **Curve List**)

Click **Apply**

(circles which represent finite element nodes will appear on ends of the curve)

Click **Curve List** box

Click on right most curve in the main viewport

(the words "Curve 3" will be added to the **Curve List**)

Click **Apply**

(circles which represent finite element nodes will appear on ends of the curve)

(The nodes created above must now be tied together with element s)

(up at the top of the RM)

Set **Action** = *Create*

Object = *Mesh*

Type = *Curve*

Click on Bar2 under **Element Topology**

Click **Curve List** Box

Click the left most curve in the main viewport (should be curve 1)

Click **Apply**

Click **Curve List** Box

Click the middle curve in the main viewport (should be curve 2)

Click **Apply**

Click **Curve List** Box

Click the right most curve in the main viewport (should be curve 3)

Click **Apply**

(numbers for the nodes will appear over the geometry points)

(up at the top of the RM)

Set **Action** = *Equivalence*

Object = *All*

Type = *Tolerance Cube*

(The purpose here is to tie the nodes together that lie on top of one another)

Set the **Equivalencing Tolerance** to *.005*

Click **Apply** (at the bottom of the RM)

(The command window at the bottom of the PATRAN desktop will tell you that 2 nodes were deleted. In addition circles will appear over the ends of the middle curve to indicate the equivalencing of the "overlapping" nodes)

The boundary conditions are specified next:

From the TM choose **Load/BC's**

A RM called **Load/Boundary Conditions** will appear

Set **Action** = *Create*

Object = *Displacement*

Type = *Nodal*

Set **Current Load Case** = *Default*

Enter **New Set Name** as

RLClamp

(This is for the right and left clamping of the bar structure)

Click **Input Data...**

a SM appears

Set **Input Translations** to $\langle 0,0,0 \rangle$

Be sure **Analysis Coordinate Frame** is *Coord0*

Click OK

(back in the **Load/Boundary Conditions** RM)

Click **Select Application Region**

Turn on the **Geometry** (button down)

Click in box under **Select Geometric Entities**

A Patran item menu appears (just to the left of the RM)

Click on the picture with a point in this menu

In the main view port, click on the left most point on the line

A SM called **Selection Choices** appears

Choose **Point 1**

(This will cause the words "Point 1" (assuming point 1 is the leftmost
point on the line) to appear in the **Select Geometric**

point on

Entities box in the RM)

Click on **Add** just below this box

(This will remove the words "Point 1" from the **Select Geometric**

Entities box and add them to the **Application Region** box)

Click in the **Select Geometric Entities** box again.

Next Click point 2 in the main view port (assuming point 2 is the right
most point in the bar structure)

most point

A SM called **Selection Choices** appears

Choose **Point 2**

Click **Add** (The **Application Region** box should now have the words

"Point1 2" in it and the **Select Geometric Entities** box should be empty)

Click OK

(The **Load / Boundary Condition** RM appears again)

Click **Apply**

(3 displacement constraint arrows should now appear in the main
viewport window on the extreme right and on the extreme left points in
the bar structure)

The loads are specified next:

(Continuing on in the **Load/BC's** RM)

change **Action** = *Create*

Object = *Force*

Type = *Nodal*

Change the **New Set Name** to *axial3*

Click **Input Data...**

a SM appears

Enter the force vector $\langle 1.8E4, 0, 0 \rangle$
leave the moments $\langle \quad \rangle$ (i.e. blank)
Click **OK**

(Continuing on in the **Load/BC's RM**)

Click **Select Application Region**

a **Select Application** menu appears as well as a small **Patran** item

menu

In the **Select Applications** menu, turn on the **Geometry**

Filter

Next, click in the box labeled **Select Geometric Entities**
Click in the **Patran** item menu (just to the left of the RM)
on the point icon

In the main viewport, click on the 3rd point from the left
(its number (should be Point 4) will be added to the **Select Geometric Entities** list)

Click **Add**

(the point's number will be added to the **Application**

Region list)

Click **OK**

(**Load/BC's** menu now reappears)

Click **Apply** (bottom of the RM)

(A vector with the load should appear on the 3rd point from the left in the
viewport)

main

The materials are specified next:

On the TM select **Materials**

a RM will appear called **Materials**

Set **Action** = *Create*

Object = *Isotropic*

Method = *Manual Input*

Click **Material Name** box

Input the name *Steel*

Click **Input Properties**

SM called **Input Options** appears

Input **Elastic Modulus** = $2.0E11$

Input **Poisson** = 0.3

Click **OK**

Back in the Materials RM, click **Apply**

Click **Material Name** box

Input the name to be *Aluminum*

Click **Input Properties** box

SM called **Input Options** appears

Input **Elastic Modulus** = $7.0E10$

Input **Poisson** = 0.3

Click **OK**

Back in the Materials RM, click **Apply**

(The **Existing Materials** box should have *Steel* and *Aluminum* in it)

The properties for each element are assigned next:

On the TM select **Properties**

a RM will appear called **Element Properties**

Set **Action** = *Create*

Dimension = *1d*

Type = *rod*

Click **Property Set Name** box

Enter *bar1*

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "Steel" in the **Materials Property Set** box

(the words m:Steel will appear in the **Material Name** box)

Click in the **Area** box

Enter *0.0004*

Click **OK**

(note: If you just input the word Steel in the **Material Name** box,
the element will not have the correct properties. The exact

syntax

m:Steel is necessary)

(Back in the **Element Properties** RM)

Click **Select Members** box

a Patran item menu will appear to the left of the RM

In the item menu, click in the box which contains the element with
end nodes (as opposed to the curve in the left box)

(This allows you to pick finite element entities as opposed to the
geometric entities in the other box)

Click on element 1 in the main viewport

(element 1 is the left most element in the bar structure)

(The words Elm 1 will appear in the **Select Members** box)

Click **Add**

(The words Element 1 appear in the **Application Region** box)

Click **Apply** in the **Element Properties** menu

(Bar 1 will be added to the **Existing Property Sets** box)

Change **Property Set Name** to bar2

Click **Input Properties...**

a SM called **Input Properties** will appear

Click the **Material Name** box

Click **Aluminum** in the **Materials Property Sets** box
(The words m:Aluminum will appear in the **Materials Name** box)
Change the **Area** to 0.0025
Click **OK**

(Back on the **Element Properties** Menu)

Click the **Select Members** box

A Patran item menu appears just to the left of the RM

In this item menu, click in the box which contains the element with end nodes (as opposed to the curve in the other box)

Click on element 2 in the main viewport

(Element 2 is the middle element in the bar structure)

(The words Elm 2 appears in the **Select Members** box)

Click **Add**

(The words Element 2 appear in the **Application Region** box)

(Note: If anything other than Element 2 is in the **Application Region** box, it must be deleted.)

Click **Apply**

(The words bar2 will be added to the **Existing Properties Sets** box)

Change **Property Set Name** to bar3

Click **Input Properties...**

a SM called **Input Properties** will appear

Click the **Material Name** box

Click **Aluminum** in the **Materials Property Sets** box

(The words m:Aluminum will appear in the **Materials Name** box)

Change the **Area** to *0.0001*

Click **OK**

Click the **Select Members** box

A Patran item menu appears just to the left of the RM

In this item menu, click in the right box which contains the element with end nodes (as opposed to the curve in the other box)

Click on element 3 in the main viewport

(Element 3 is the right most element in the bar structure)

(The words Elm 3 appears in the **Select Members** box)

Click **Add**

(The words Element 3 appear in the **Application Region** box)

(Note: If anything other than Element 3 is in the **Application Region** box, it must be deleted.)

Click **Apply**

(The words bar3 will be added to the **Existing Properties Sets** box)

The analysis is to be done is specified next:

On the TM select **Analysis**

a RM will appear called **Analysis**

Set **Action** = *Analyze*

Object = *Entire Model*

Method = *Full Run*

Click on **Translation Parameters**

a SM will appear

Change the **Data Output** to **OP2 and Print**

Click **OK**

Click on **Solution Type**

a SM will appear

Set **Solution Type** = **Linear Static** (button down)

Click **OK**

(back in the analysis menu)

Click **Apply**

(The analysis will take a few seconds [maybe 10] to run)

In the RM analysis

Set **Action** = *Read Output 2*

Object = *Result Entities*

Method = *Translate*

Click on **Select Results File**

a SM will appear

Find and select the file **bar.op2**

(You may need to use the “find” tools in Windows to locate the file.

Occasionally Nastran will put the *.op2 file in a weird place.

Occasionally it even puts the file on the hard drive of the license file

server. If you cannot find the file on your local hard drive then look on

the file servers hard drive. The file server for the NCL is DFELAB10.

The file server for the library is HOPPER. You should be able to access

either of these from your local machine over the network)

Click **OK**

Back in the **Analysis** RM

Click **Apply**

Next you will post process the results by viewing and exporting them

On the TM select **Results**

a RM will appear called **Results**

Set **Action** = *Create*

Object = Quick Plot

A SM appears

Under **Select Result Case**

highlight the option *Default, Static Subcase*

Under **Select Fringe Result**

Highlight *Displacements, Translational*

Under **Select Deformation Result**

Highlight *Displacements, Translational*

Click **Apply**

A Colored picture displaying the displacement results will appear. It includes numeric results for max and min displacement as well as color-coded results for the entire beam.

To save this plot use the “copy to Clipboard” icon (usually just to the right of the print icon) to copy the viewport to the clipboard. Then paste the picture into a word processing document.

If you want to print the viewport directly, you can just use the normal Windows commands (File/Print)

Next, to see the stresses

Under **Select Result Case**

Highlight the option *Default, Static Subcase*

Under **Select Fringe Result**

Highlight *Stress, tensor*

Change the **Quantity** to *X Component*

Under **Select Deformation Result**

Highlight *Displacements, Translational*

Click **Apply**

A Colored picture displaying the stresses results will appear. It includes numeric results for max and min Stresses as well as color-coded results for the entire beam.

To save this plot use the “copy to Clipboard” icon (usually just to the right of the print icon) to copy the viewport to the clipboard. Then paste the picture into a word document.

If you want to print the viewport directly, you can just use the normal Windows commands (File/Print)

Next you will end your PATRAN session by saving your database and exiting

On the TM select **File**

From the pull down menu select **Save**

On the TM select **File**

From the pull down menu select **Quit**

VI. EXERCISES:

1. Hand in the output file bar.f06. In this file, highlight the reaction forces, stresses and the displacements.
2. Hand in the two picture files which have the pictures of your finite element model and the displacement and stress results.
3. Are any of the members in or close to the plastic range of the material?
4. Check the problem against some analytic answer to see if your displacement and stress results are the correct order of magnitude. It might be easiest to solve the statically determinant problem and use that as a bound for the displacements and stresses as opposed to solving the statically indeterminate problem. If you decide to use this approach, explain how the statically determinant problem gives bounds for the displacements and stresses. Are these upper or lower bounds? Are your FEA based answers consistent with this analytic check?
5. Will it increase the accuracy of the results to use a greater number of elements? Why or why not?
6. Are there any physical phenomena that this bar might experience that we have not taken into account?
7. Will this type of element correctly capture the physics of the problem if the lower force is set to zero and the upper force is maintained at 9000 N? Why or Why not?

MSC Patran Tutorial # 2 Modeling of a Truss

I. THE PHYSICAL PROBLEM:

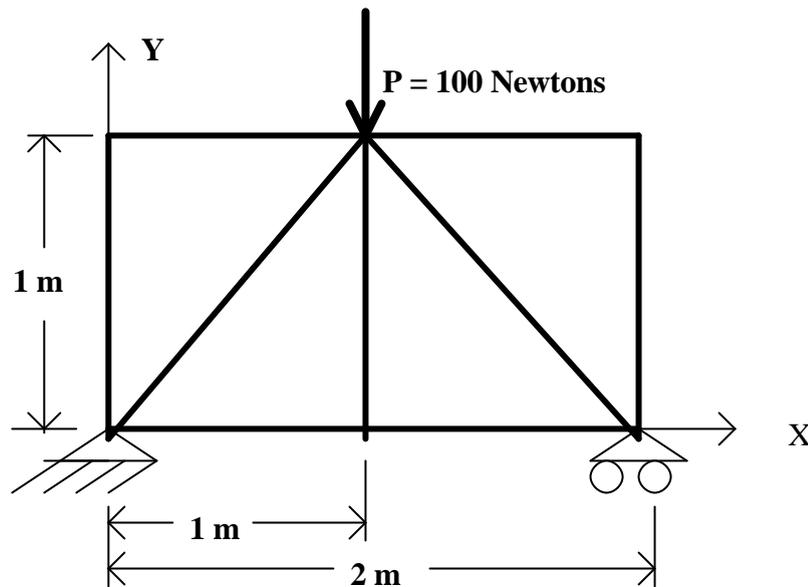
The truss structure shown below has nine members. Each of the members is made of aluminum and each has the same cross sectional area. The lower left corner of the structure is constrained in all three directions. The lower right hand corner is constrained in the Y and Z directions, but is free to roll in the X direction. A vertical load of 100 Newtons is applied at the midpoint of the top of the truss. The loading is directed downward. The truss geometry is symmetric about the vertical line through the point at which the force is applied. Material properties, as well as physical dimensions, are given below.

For the truss below:

Young's modulus = $70 \times 10^9 \text{ N}/(\text{m}^2)$ (Aluminum)

Poisson's ratio = 0.3

Truss members are (3 cm X 3 cm) square



II. THINKING ABOUT THE MECHANICS

Before you begin the computational model of the structure, study the structure for a few minutes to determine if it has any peculiarities. Ask a few introductory questions:

- I. Is the truss constrained from any rigid body displacements or rotations?
- II. What direction do you expect the reactions to be in?
- III. What magnitudes should the directions have?
- IV. Are there any “zero-force” members in the truss?
- V. Use the method of sections (or another method if you prefer) to analytically determine the stresses in a few of the members.

III. THE GEOMETRIC AND FINITE ELEMENT MODEL

In the modeling instructions below, the geometry is specified by creating the MSC/Patran geometric entity called a "curve" between each of the truss's joints. In this manner, each truss member becomes a separate curve in the geometric portion of the database. The lengths and directions of the curves correspond to those of the members in the physical truss structure.

Each of the truss members is modeled using a single 2-node rod element. Each element is originally created with two unique nodes which no other element shares. The procedure called "equivalencing" in MSC/PATRAN creates a single node from two or more nodes which have the same physical location. Therefore, after equivalencing, there are nine elements and six nodes in this structure. These elements have three displacement degrees of freedom per node. The elements can only model axial (membrane) deformations. Bending type deformations, which are evidenced by rotation of the element cross section, are not accounted for by this particular element. Torsion of the members is also neglected. The neglect of torsion and bending are very common assumptions in truss problems, as these are higher order effects in a great number of truss type structures. Physically, this non-bending assumption is representative of pinned joints (for 2-D) or spherical joints (for 3-D). It should be noted, however, that there are some situations where these assumptions would not allow your model to correctly capture the physics of the problem. This type of modeling assumption should be carefully considered.

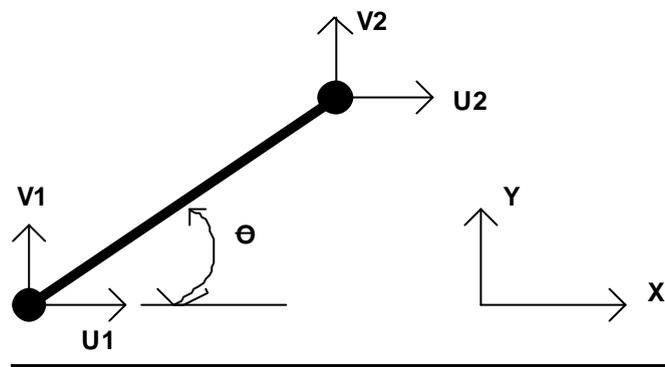
The loading is modeled with a single concentrated force of magnitude 100 on the center node of the top of the structure. It is also possible to position loads on geometric entities like points and surfaces instead of on finite element entities like nodes. This is demonstrated in other tutorials. The boundary conditions are established by constraining the displacements at the lower left node to be zero in all 3 directions and the lower right node to be zero in the Y and Z directions. Material properties and lengths are input corresponding to the figure of the truss above. Note that it is not necessary to carefully number the nodes of the structure for minimization of the bandwidth of the stiffness matrix. The code automatically renumbers the nodes for bandwidth minimization before solving the system of equations.

IV. THE FINITE ELEMENT THEORY

The finite elements used to model two and three dimensional truss structures are actually just the simple 2-node bar elements spatially extrapolated to function in two or three dimensional space. This spatial extrapolation is in the form of a transformation of the axial direction of the arbitrarily oriented bar into the global (fixed) coordinate system. The results of the transformation is found in the following stiffness matrix for the two dimensional case.

$$K = \frac{A E}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix}$$

where the order of the degrees of freedom is $\{u_1, v_1, u_2, v_2\}$. The A, E, and L are the cross sectional area, Young's (elastic) modulus and axial length respectively. The c and s in the matrix stand for Cos (θ) and Sin (θ) respectively. The orientation of the bar and the angle θ are shown below.



This element does not have any stiffness associated with rotational degrees of freedom. Therefore, bending and torsion effects are not included in this model nor is it possible to load the structure with moments. Also, the element, in the manner it is used in this analysis, does not have the ability to model large deformations and will not warn the user in case of buckling type failures (i.e. geometric nonlinearities). Similarly, this type of analysis does not have the ability to correctly model stresses which are not in the elastic range of the material (i.e. material nonlinearities).

V. STEP BY STEP INSTRUCTIONS FOR BUILDING THE TRUSS MODEL USING PATRAN

Preliminaries for using MSC/PATRAN include:

- 1) Log on to the computer
- 2) Change to the directory that you wish to contain your analysis results
- 3) Left click START (lower left corner of the NT desktop), go to PROGRAMS, then top MSC (common), then to MSC Patran 90. This will bring up the MSC/Patran Program.

In the instructions below, the following abbreviations and terms will be used:

TM = Top Menu. This refers to the horizontal menu options residing at the top of the screen after PATRAN has been initiated.

RM = Right Menu. This refers to the menus that pop up after an option has been chosen from the top menu. These menus reside on the far right side of the PATRAN desktop.

SM = Subordinate Menu. This refers to the menus that pop up from options selected in the right menu.

Click = Unless otherwise stated, this indicates a click with the left mouse button.

Boldface will indicate text that occurs in the PATRAN menus.

Italics text will indicate text that you must enter into text boxes in the PATRAN menus or text that you choose in a menu scroll box.

1. Our first step is to create a new database:

From the TM choose **File**

In the resulting pull down menu choose **New Database**

A SM called **New Database** pops up

Turn off (button up) **Modify Preferences**

Under **New Database Name** enter *truss.db*

Click **OK**

A menu called **New Model Preferences** will appear

Select **Tolerance** to be **based on the model**

Set **Model Dimension** = *2.0*

Analysis code = *MSC/Nastran*

Analysis Type = *structural*

Click **OK**

2. The geometry of the truss will be determined next:

From the TM choose **Geometry**

A RM called **Geometry** will result

Set **Action** = *Create*

Object = *Curve*

Method = *XYZ*

Set the **Curve ID** list to *1*

Set **Reference Coordinate Frame** to *Coord 0*

Turn off the **Auto Execute** button (uncheck)

Enter the following into the **Vector Coordinates** list:

<1,0,0>

Enter the following into the **Origin Coordinates** list:

<0,0,0>

(note that PATRAN will accept either commas or blanks as separators between coordinates)

Click **Apply**

Build the rest of the truss using the following table

| Line Number | Vector Coordinates | Origin Coordinates |
|-------------|--------------------|--------------------|
| 1 | <1,0,0> | <0,0,0> |
| 2 | <0,1,0> | <0,0,0> |
| 3 | <0,1,0> | <1,0,0> |
| 4 | <1,0,0> | <0,1,0> |
| 5 | <1,1,0> | <0,0,0> |
| 6 | <1,0,0> | <1,0,0> |
| 7 | <1,-1,0> | <1,1,0> |
| 8 | <1,0,0> | <1,1,0> |
| 9 | <0,1,0> | <2,0,0> |

Note that the commands Construct, Line, XYZ do NOT work based on the coordinates of the 2 end points of the truss member. These commands generate lines based on the origin and the vector for that particular truss member.

Note that if you make a mistake you can erase by clicking on the undo button on the top of the PATRAN desktop. This will erase the LAST CONSTRUCTION COMMAND ONLY. In other words, it will take the process back to before you hit the **Apply** button the last time.

3. The boundary conditions are specified next:

From the TM choose **Load/BC's**

A RM called **Load/Boundary Conditions** will appear

Set **Action** = *Create*

Object = *Displacement*

Type = *Nodal*

Set **Current Load Case** = *Default*

Enter **New Set Name** as *leftfix*

(This is for the clamping of the left most bottom nodes)

Click **Input Data...**

a SM appears

Set **Load/BC Scale Factor** = *1.*

Set **Translations** to *<0,0,0>*

Leave the **Rotations** blank

Be sure **Analysis Coordinate Frame** is *Coord0*

Click OK

(back in the **Load/Boundary Conditions** RM)

Click **Select Application Region**

a SM called **Select Application Region** appears with a Select menu on its left edge.

In the **Select Application Region** SM

Turn on the **Geometry** (button down)

Click in box under **Select Geometric Entities**
 In the Select Menu (which is just to the left of the SM)
 Click on the picture with a point
 In the main view port, click on point 1 (left most point on the
 bottom edge)
 A Selection Choices menu will appear. Choose Point 1.

 (This will cause the words "Point 1" to appear in the **Select**
Geometric
Entities box in the RM)
 Click on **Add** just below this box
 (This will remove the words "Point 1" from the **Select Geometric**
Entities box and add them to the **Application Region** box)
 Click OK
 (The **Load / Boundary Condition** RM appears again)
 Click **Apply**

(3 displacement constraint arrows and the numbers 1,2,3 should now appear in the main viewport window on the extreme right point on the bottom of the truss)

Back in the RM called **Load/Boundary Conditions**

Set **Action** = *Create*

Object = *Displacement*

Type = *Nodal*

Set **Current Load Case** = *Default*

Enter **New Set Name** as *rightfix*

(This is for the clamping of the right most bottom nodes)

Click **Input Data...**

a SM appears

Set **Load/BC Scale Factor** = *1.*

Set **Translations** to $\langle ,0,0 \rangle$

Note the space left in before the first comma in the **Translations** vector. This ensures that the X direction is NOT constrained

Leave the **Rotations** blank

Be sure **Analysis Coordinate Frame** is *Coord0*

Click OK

(back in the **Load/Boundary Conditions** RM)

Click **Select Application Region**

a SM called **Select Application Region** appears with a Select menu on its left edge.

In the **Select Application Region** SM

Turn on the **Geometry** (button down)

Click in box under **Select Geometric Entities**

In the Select Menu (which is just to the left of the SM)
Click on the picture with a point
In the main view port, click on point 5 (right most point on the
bottom edge)

A Selection Choices menu will appear. Choose Point 5.

(This will cause the words "Point 5" to appear in the **Select**

Geometric

Entities box in the RM)

Click on **Add** just below this box

(This will remove the words "Point 5" from the **Select Geometric**

Entities box and add them to the **Application Region** box)

Click **OK**

(The **Load / Boundary Condition** RM appears again)

Click **Apply**

(2 displacement constraint arrows and the numbers 2,3 should now appear in the main viewport window on the extreme right point on the bottom of the truss)

4. The loads are specified next:

(Continuing on in the **Load/BC's** RM)

change **Action** = *Create*

Object = *Force*

Type = *Nodal*

Change the **New Set Name** to *topload*

Click **Input Data...**

a SM appears

Enter the force vector $\langle 0, -100, 0 \rangle$

leave the moments $\langle \quad \rangle$ (i.e. blank)

Click **OK**

(Continuing on in the **Load/BC's** RM)

Click **Select Application Region**

A SM called **Select Application Region** appears with a select menu just
to its left

In the **Select Application Region** menu

Select the **Geometry Filter** = *Geometry*

Click in the **Select Geometry Entities** box

In the select menu to the left of the SM

Click on the point icon

In the main viewport, click on the point 4 (top center point)

(point 4 will be added to the **Select Geometric Entities** list)

In the **Select Application Region** menu

Click **Add**

(Point 4 will be added to the **Application Region** list)

Click **OK**

(**Load/BC's** menu now reappears)

Click **Apply**

(A vector with the load of magnitude 100 in the $-Y$ direction will appear on point 4 in the main viewport)

5. The finite element mesh is specified next:

From the TM choose **Elements**

A RM appears called **Finite Elements**

Set **Action** = *Create*

Object = *Mesh Seed*

Type = *Uniform*

Select **Number of Elements** (button down)

Number = 1

Turn off the **Auto Execute** (button up)

Click in **Curves List** box

Click on curve 1 in the main viewport

(curve 1 is the line between point 1 and point 2. This is the bottom left part of the truss)

(The words "Curve 1" will be added to the **Curve List**)

Click **Apply**

(circles which represent finite element nodes will appear on points 1 and 2)

Do the same for curves 2-9.

(The nodes created above must now be tied together with elements)

(up at the top of the RM)

Set **Action** = *Create*

Object = *Mesh*

Type = *Curve*

Click on Bar2 under **Element Topology**

Click **Curve List** Box

Click curve 1 in the main viewport

Click **Apply**

Do the same for curves 2-9

To see the element numbers on the truss, click the "Label Control" button (Looks like an "L") on the top row menu. This adds a label control tool bar which allows you to turn on/off labels for different geometric and/or finite element entities.

(up at the top of the RM)

Set **Action** = *Equivalence*

Object = *All*

Type = Tolerance Cube

(The purpose here is to tie the nodes together that lie on top of one another)

Leave the **Nodes to be Excluded** list blank

Set the **Equivalencing Tolerance** to *.001*

Click **Apply**

(The command window at the bottom of the PATRAN desktop will tell you that 12 nodes were deleted)

6. The materials are specified next:

On the TM select **Materials**

a RM will appear called **Materials**

Set **Action** = *Create*

Object = *Isotropic*

Method = *Manual Input*

Click **Material Name** box

Input the name to be *Aluminum*

Click **Input Properties** box

SM called **Input Options** appears

Input **Elastic Modulus** = *7.0E10*

Input **Poisson** = *0.3*

OK

Back in the Materials RM

Click **Apply**

(The **Existing Materials** box should have *Aluminum* in it)

7. The properties for each element are assigned next:

On the TM select **Element Properties**

a RM will appear called **Element Properties**

Set **Action** = *Create*

Dimension = *1d*

Type = *Rod*

Click **Property Set Name** box

Enter *truss1*

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "Aluminum" in the **Materials Property Set** box

(the words m:Aluminum will appear in the **Material Name** box)

Click in the **Area** box

Enter *.0009* (recall that the member's cross section was 3cm x 3cm square)

Click **OK**

(Back in the **Element Properties** RM)

Click **Select Members** box

In the select menu just to the left of the SM

Click in the box which contains finite element with 2 end nodes
(This allows you to pick finite element entities as opposed to
the geometric entities in the other box)

Move the cursor arrow to a point to the left and above the highest, left-
most point on the truss. Click and hold down the left mouse button.

Drag the cursor (while holding down the mouse button) to a
point to the right of and below the right-most bottom node. A
"selection box" is formed while you drag. Release the button.

(The words Elm 1:9 will appear in the **Select Members** box)

Click **Add**

(The words Element 1:9 appears in the **Application Region** box)

Click **Apply** in the **Element Properties** menu

(truss1 will be added to the **Existing Property Sets** box)

8. The analysis is to be done is specified next:

On the TM select **Analysis**

a RM will appear called **Analysis**

Set **Action** = *Analyze*

Object = *Entire Model*

Method = *Full Run*

Click on **Solution Type**

a SM will appear

Click on **Translation Parameters**

A SM called **Translation Parameters** will appear

Set **Data Output** to *OP2 and Print*

Click **OK**

Back in the **Analysis** RM

Set **Solution Type** = **Static** (button down)

Click **OK**

(back in the RM **Analysis**)

Click **Apply**

(The analysis will take a few seconds to run)

Now we'll read the results into the graphics database

(back in the RM **Analysis**)

Set **Action** = *Read Output2*

Object = *Result Entities*

Method = *Translate*

Click on **Select Results File**

Choose *truss.op2* (you may need to go to the root or home directory to find this. If this file does not exist, then there was an error in your model.

Go to the file *truss.log* or *truss.f06* to attempt to find out what error occurred.)

Back in the **Analysis** RM

Click **Apply**

9. Visualize the results

From the TM choose **Results**

A RM called **Results** appears

Set Action = Create

Object = Quick Plot

Under **Select Fringe Result** Choose *Displacements, Translational*

Set **Quantity** = *Y Component*

Under **Select Deformation Results**, choose *Displacements, Translational*

Click **Apply**

(A deformed plot appears with colors indicating the level of deformation. Note that the visual deformation of the truss is magnified so that you can see the deformation “mode”. The actual truss deformations are very small; as can be seen by the numerical values, which are NOT scaled)

Note that you can also view the stress results in this manner. Simply choose **Stress, Tensor** from the **Select Fringe Result** options. Recall that there are a number of ways to compute and extrapolate the stresses for a bar and these will make significant differences in the values which are plotted.

10. Check the written report of the truss results.

The file containing the written results from the analysis is *scaled truss.f06*. Open the file (by simply double clicking on it). The file might be in the root or home directory or in the directory from which you ran the analysis.

In this file find the displacement vectors and record the numerical values. These will help you answer some of the question below. Also, find the vectors for the stresses and constraint forces and record these values.

Next you will end your MSC PATRAN session by saving your database and exiting

On the TM select **File**

From the pull down menu select **Save**

On the TM select **File**

From the pull down menu select **Quit**

VII. QUESTIONS FROM THE TUTORIAL: MODELING A TRUSS

The questions below refer to the truss model described at the beginning of this tutorial. Also, information from the output file truss.f06 will be needed in order to answer many of these questions. As used below, the term "**member**" refers to the portion of a truss structure between two joints. For example, the top of this structure has two horizontal members which are connected by the joint at which the load is applied.

- 1a.** What is the maximum displacement for the structure ?
- 1b.** Is this displacement consistent in location, magnitude and direction with your physical intuition ?
- 2a.** What is the maximum stress in the structure ?
- 2b.** Is this stress consistent in location, magnitude and direction with your physical intuition ?
- 3.** Are there any members with very low stresses? Does this make physical sense?
- 4.** How many equations are solved in order to determine the displacements for this structure ?
- 5.** What assumptions are involved in using this specific element as opposed to using a 2 node beam element with 6 degrees of freedom (3 displacements and 3 rotations) per node ?
- 6.** The present model uses a single 2-node bar element for each truss member. Would the accuracy of the model increase if two bar elements were used to model each truss member ? Justify your answer.
- 7a.** The resultant forces (sometimes called constraint, restoring or reaction forces), are located at the nodes where the boundary conditions are applied. State how these resultant forces can be used as a "necessary but not sufficient" test of the accuracy of your analysis.
- 7b.** Does your analysis pass this test ?
- 8.** If two nodes in your final truss structure have the exact same physical location but different node numbers, what part of the PATRAN analysis procedure has been left out ?
- 9a.** How could the element properties be changed to model this truss if the members in the structure were circular hollow aluminum bars. Assume that the outside diameter is 3 cm and the inside diameter is 2 cm. Remember that this structure only models the membrane (axial) deformation not the bending deformation of each member.
- 9b.** If you wanted to account for bending deformation in your model, could you use this same adjustment to the physical properties to model the truss with hollow members ?
- 10.** Assume that the cross sectional area of the truss members is incorrectly input in square cm as opposed to square meters. If the other data for the problem is input using meters, what would the maximum deflection of the truss be ?
- 11.** Assuming that the rotations of the cross sections of the bars are small, what will be the difference between the results of your PATRAN analysis and the exact analysis ? ("exact" here refers to the analytic analysis using standard structural analysis methods)
- 12a.** Some truss structures may be designed so that, if certain members of the truss are damaged to the extent that they no longer have significant stiffness, the structure will still

be able to handle reasonable loading. This type of truss assembly is said to have redundant members. Without changing the number of elements in the structure, suggest a method of using MSC PATRAN to determine if there are redundant members in this truss structure.

12b. Use the method developed in 13a) to determine if one of the diagonal members is redundant.

12c. Use the method developed in 13a) to determine if one of the vertical members is redundant.

13a. Predict the deflection if the direction of the load is changed from the negative Y direction, to the Z direction (note from your nodal location information that this truss is located in the X - Y plane).

13b. Run the analysis and explain the displacement results.

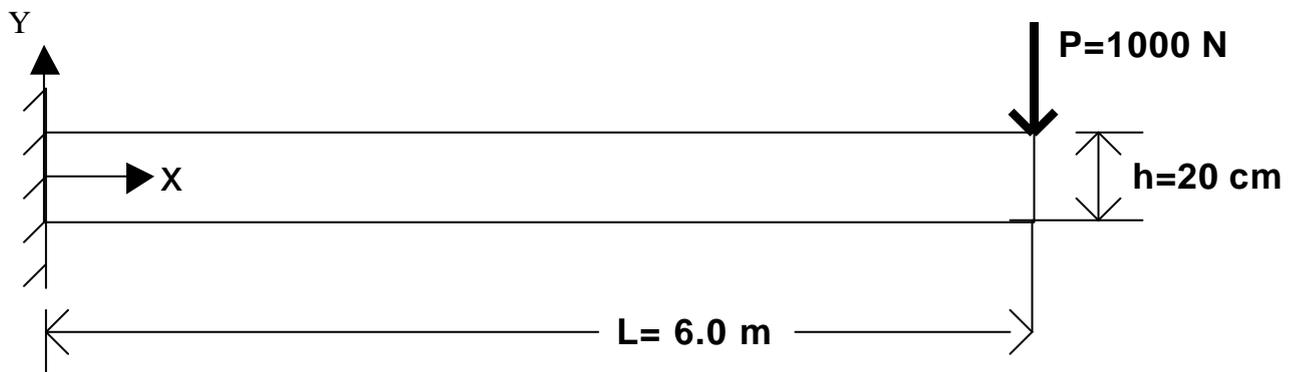
14a. Predict the effect of removing the displacement boundary condition on the lower right node of the truss structure ?

14b. Run the analysis and explain the displacement results.

MSC/PATRAN TUTORIAL # 3
MODELING A CANTILEVERED BEAM WITH END LOAD
USING 4 NODE SHELL ELEMENTS

I. THE PHYSICAL PROBLEM

The beam below is cantilevered or "built in" on the left edge. This means that both the translations and the rotations are held to zero along this edge. A point or concentrated load of magnitude 1000 N (approximately 225 lb) in the negative Y direction is found at the tip of the beam. This problem is part of a standard set of test cases for finite elements published in a paper by MacNeal and Harder (MacNeal founded the company that makes the FEA code MSC/NASTRAN and MSC/PATRAN). The set of problems is called "The MacNeal - Harder Test Cases". The material properties for the beam are $E = 200 \times 10^9$ Pascals (typical for steel) and $\nu = 0.0$ (as the analytic beam theory we use below does not take Poisson's ratio effects into account). The beam has a solid rectangular cross section with thickness in the Z-direction $t = 0.1$ meters and height in the Y-direction $h = 0.2$ meters.



II. THINKING ABOUT THE MECHANICS

The analytic solution for stresses and displacements for this problem is readily available. Any Mechanics of Materials text will provide equations for the max stress (located at the built in edge and on either the top fiber for max tensile stress or the bottom fiber for max compressive stress) and the max displacement (located, of course, at the free tip where the load is applied). These equations are given below.

For the normal stress due to bending:

$$s_{xx}(x) = \frac{M(x)y}{I} \text{ so that the max value located at the built-in edge is}$$

$$s_{xx-MAX} = \frac{PL(\frac{h}{2})}{\frac{1}{12}bh^3}.$$

For the displacement at the tip of the beam (maximum displacement):

$$d_y(x=L) = \frac{PL^3}{3EI}$$

Some basic questions to consider before creating the computational model are:

- Where will the stresses be tensile and where will they be compressive?
- What will be the magnitude and direction of the reaction forces/moments?
- Where will the stresses be zero?
- How do the displacements vary along the length (linear, quadratic etc.)?
- What will the local effect of the concentrated load be on the stresses?
- Is the model fully constrained from rigid body rotations and displacements?

Answering these questions qualitatively, along with the quantitative analytical solutions for the max stress and displacement will provide reinforcement that your computational model is correctly constructed.

III. GEOMETRIC AND FINITE ELEMENT MODEL

As is the standard procedure for building MSC/Patran models, we will build the geometry first and then construct a finite element mesh on that geometry. The geometry will proceed from creation of points to lines to surfaces for this simple model. Next, we will use 4 node shell elements deforming in their membrane mode to model the beam. In this exercise, we will vary the exact number and configuration of these elements. This is discussed in detail in the next paragraph. Next, the material and element properties will be entered. We will constrain the 3 displacement and 3 rotational degrees of freedom on the left edge (for both nodes). This creates the cantilevered or built-in, end condition. Then we will, place a point load of magnitude 1000 on the top right node of the tip (or right-most) element. This load will be in the negative Y direction. Finally, the nodes must be equivalenced before the analysis is ready to run.

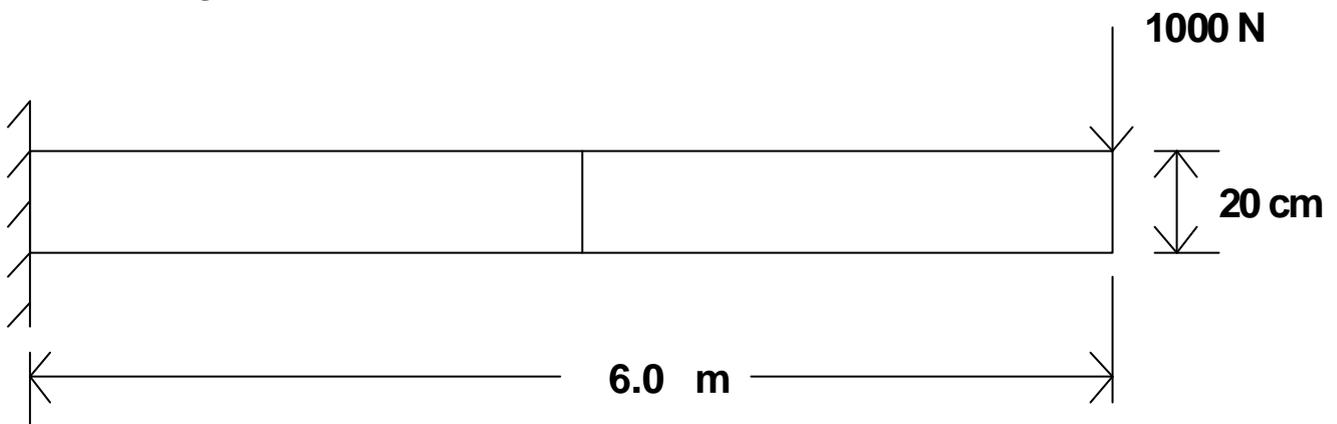
Below, we show 5 mesh configurations for the beam (labeled “a” through “e”).

Comparison of results between mesh “a” and mesh “b” will indicate of how the number of elements affects the model’s ability to correctly model a beam problem. Increasing the number of elements in a mesh in order to increase the accuracy of the results is called “h” convergence. Meshes “b” – “e” all have 6 elements; but the elements have different orientations. Elements that have non-regular shapes are said to be distorted. Distorted elements can cause errors in the FEA results. This can be a significant problem in

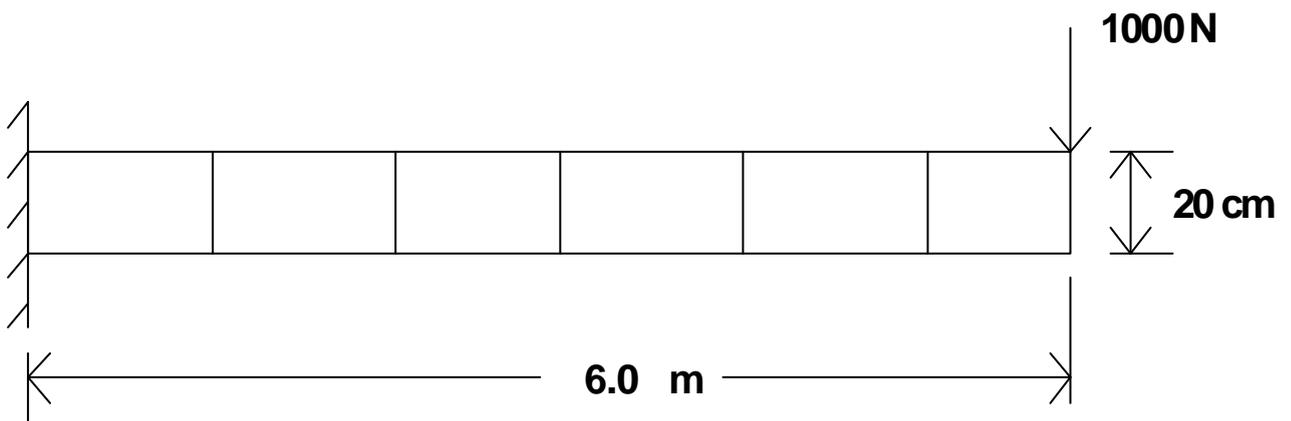
complex meshes as even the best automatic mesh generators often produce some distorted elements. The elements in MSC/Nastran have been specifically designed to minimize this unfortunate effect, but some sensitivity to element distortion may still remain. Different types of element distortion result in different levels of error. Evaluating results from the meshes “b” - “e” will provide you with some feel for how these elements perform when they are distorted.

Meshes for the “h” Convergence & Distortion Analysis

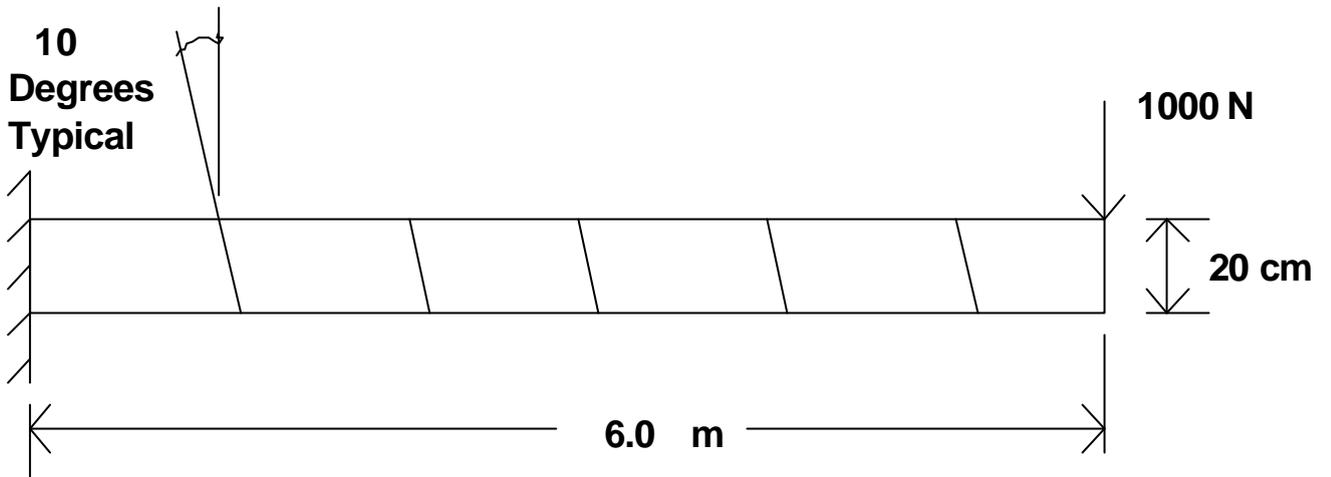
I. Rectangular 2 Element Mesh:



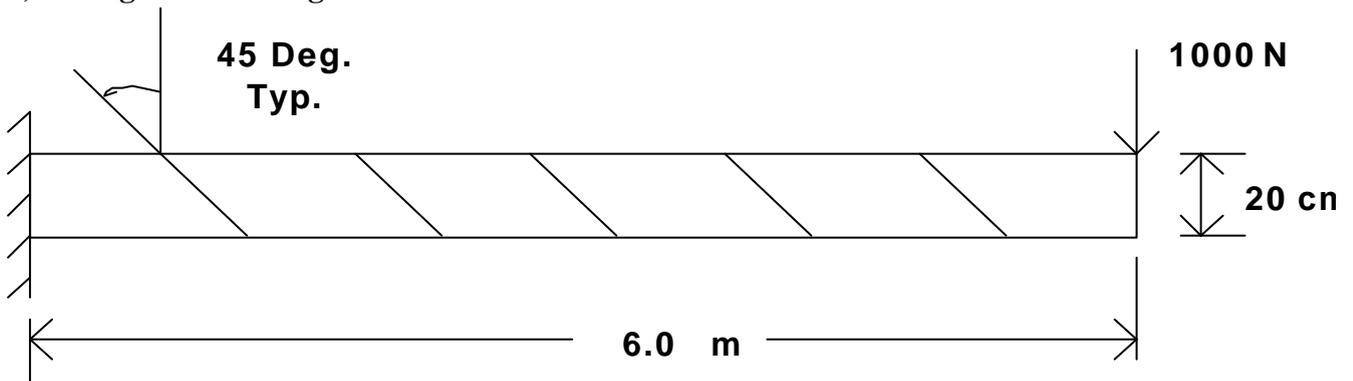
b) Rectangular 6 Element Mesh:



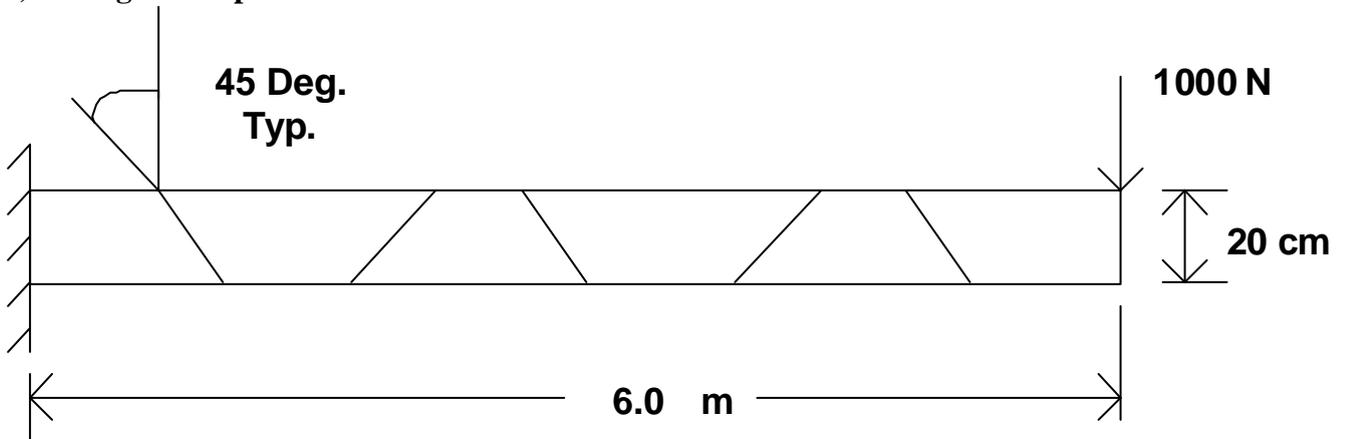
c) 10 Degree Parallelogram Mesh:



d) 45 Degree Parallelogram Mesh:



e) 45 Degree Trapezoid Mesh:

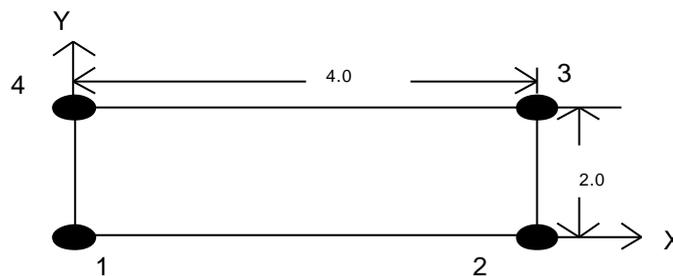


IV. FINITE ELEMENT THEORY

The exact details of the formulation of the 4 node shell elements in MSC/Nastran is rather complicated. However, the basic formulation of an isoparametric 4 node membrane element is not extremely difficult and will provide us with sufficient background information to begin to understand the “h” convergence and distortion sensitivity studies. This basic form is constructed as follows:

Isoparametric Formulation of a 2-D Membrane Element [K] Matrix

Assume the element has the configuration shown below:



The physical and natural coordinate locations of the 4 nodes are:

| NODE | (x,y) | (ξ, η) |
|------|-------|-----------------|
| 1 | (0,0) | (-1,-1) |
| 2 | (4,0) | (1,-1) |
| 3 | (4,2) | (1,1) |
| 4 | (0,2) | (-1,1) |

Our goal is to find the element stiffness matrix

$$\text{ASSUME: } [K] = \int_V [B]^T [E] [B] dV$$

ASSUME: 2 displacement degrees of freedom (dof) per node

With : [B] = the strain - displacement matrix such that $[B]\{u\} = \{e\}$

where: $\{u\}$ is the dof vector and $\{e\}$ is the strain vector
 $[E] =$ the constitutive matrix such that $[E]\{e\} = \{s\}$
 where $\{s\}$ is the stress vector and
 $V =$ volume.

Step 1: Interpolate the dof: $\begin{Bmatrix} u \\ v \end{Bmatrix} \approx [N] \{u_1, v_1, u_2, v_2, u_3, v_3, u_4, v_4\}^T = [N]\{u\}$

where $[N]$ is the shape function matrix $[N] = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix}$

and the rules for the shape functions are : 1) N_i must be =1 at node "i"
 2) N_i must be =0 at any node not = "i"

This leads to the shape functions: $N_1 = \frac{1}{4}(1-x)(1-h)$; $N_2 = \frac{1}{4}(1+x)(1-h)$;
 $N_3 = \frac{1}{4}(1+x)(1+h)$; $N_4 = \frac{1}{4}(1-x)(1+h)$

Step 2: Find the $[B]$ matrix:

Relevant strains are $\{e\} = \begin{Bmatrix} e_{xx} \\ e_{yy} \\ g_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{1}{l_x} & 0 \\ 0 & \frac{1}{l_y} \\ \frac{1}{l_y} & \frac{1}{l_x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} = [D] \begin{Bmatrix} u \\ v \end{Bmatrix}$; but from step 1

$$\begin{Bmatrix} u \\ v \end{Bmatrix} \approx [N]\{u\}$$

So $\{e\} \approx [D][N]\{u\} = [B]\{u\}$ with $[B] = [D][N]$
 Therefore,

$[B] = \begin{bmatrix} N_{1,x} & 0 & N_{2,x} & 0 & N_{3,x} & 0 & N_{4,x} & 0 \\ 0 & N_{1,y} & 0 & N_{2,y} & 0 & N_{3,y} & 0 & N_{4,y} \\ N_{1,y} & N_{1,x} & N_{2,y} & N_{2,x} & N_{3,y} & N_{3,x} & N_{4,y} & N_{4,x} \end{bmatrix}$ where the commas denote partial differentiation.

Step 3: Use the Jacobian to find derivatives:

Isoparametric Assumption: $\begin{Bmatrix} x \\ y \end{Bmatrix} = [N]\{x_1, y_1, x_2, y_2, x_3, y_3, x_4, y_4\}^T$

i.e. the isoparametric assumption is that geometry can be interpolated using the same interpolation functions as the displacements.

$$\text{The Jacobian matrix } [J] = \begin{bmatrix} \frac{\eta_x}{\eta_{\mathbf{x}}} & \frac{\eta_y}{\eta_{\mathbf{x}}} \\ \frac{\eta_x}{\eta_{\mathbf{h}}} & \frac{\eta_y}{\eta_{\mathbf{h}}} \end{bmatrix} \approx \begin{bmatrix} N_{1,x} & N_{2,x} & N_{3,x} & N_{4,x} \\ N_{1,h} & N_{2,h} & N_{3,h} & N_{4,h} \end{bmatrix} \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ x_3 & y_3 \\ x_4 & y_4 \end{bmatrix}$$

$$\text{and from chain rule } \begin{Bmatrix} N_{i,x} \\ N_{i,y} \end{Bmatrix} = \begin{bmatrix} \mathbf{x}_{,x} & \mathbf{h}_{,x} \\ \mathbf{x}_{,y} & \mathbf{h}_{,y} \end{bmatrix} \begin{Bmatrix} N_{i,x} \\ N_{i,h} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} N_{i,x} \\ N_{i,h} \end{Bmatrix}$$

$$\text{So in this particular case: } [J] = \frac{1}{4} \begin{bmatrix} -1+\mathbf{h} & 1-\mathbf{h} & 1+\mathbf{h} & -1-\mathbf{h} \\ -1+\mathbf{x} & -1-\mathbf{x} & 1+\mathbf{x} & 1-\mathbf{x} \end{bmatrix} \begin{bmatrix} 0 & 0 \\ 4 & 0 \\ 4 & 2 \\ 0 & 2 \end{bmatrix} =$$

$$\frac{1}{4} \begin{bmatrix} 8 & 0 \\ 0 & 4 \end{bmatrix} = \begin{bmatrix} 2 & 0 \\ 0 & 1 \end{bmatrix}$$

$$\text{which implies that } [J]^{-1} = \begin{bmatrix} \frac{1}{2} & 0 \\ 0 & 1 \end{bmatrix}$$

This allows us to find the entries in [B]

Step 4: Perform the numerical integration:

Assume that the element has constant thickness = t implies $[K] = t \int_A [B]^T [E] [B] dx dy$

Which, according to the rules of calculus can be written: $[K] = t \int [B]^T [E] [B] |J| dx dh$

where $|J|$ is the determinant of the Jacobian matrix.

Gaussian numerical integration is then used to find the final numbers for the element stiffness.

This takes the form: $[K] = h \sum_{j=1}^{ng_j} \sum_{i=1}^{ng_i} [B]^T [E] [B] |J| w_i w_j |_{(\mathbf{x}_i, \mathbf{h}_j)}$

Where ng_j and ng_i are the number of gaussian integration points in the “j” and “i” directions respectively and w_j and w_i are the associated gaussian weighting factors.

Understanding the “h” Convergence Experiment:

From step 1 above we gain insight into the “h” convergence study. Remember that the that the analytic formula for the displacements as a function X (distance from built-in

edge) is: $d_y(x) = \frac{Px^2(3L-x)}{6EI}$ where P is the load, L is the length, E is the Elastic

Modulus and I is the bending moment in inertia. This equation shows that the displacement is a cubic function of the distance from the cantilever. As the bi-linear (linear in both x and h) shape functions are used to interpolate the displacements for this 4 node element, the elements are attempting to capture a cubic behavior by using a series of linear approximations. The number of linear approximations is equal to the number of elements we use (the actual situation when using MSC/Nastran’s 4 node shell element is a little better than this due to the innovative element formulation, but this is a good way to conceptually grasp the idea of “h” convergence). This is the reason why 2 elements give a higher error than do 6 elements.

Understanding the Distortion Sensitivity Experiment:

When an element is rectangular, its Jacobian matrix (used in steps 3 and 4 above) is numerically exact. However, if the element becomes distorted, the bi-linear shape functions used to form [J] can no longer exactly capture the geometry and the Jacobian is no longer numerically exact. This introduces error into steps 3 and 4 above. The exact form of the element’s distortion determines the amount of error which is introduced. As mentioned previously, the elements in MSC/Nastran are intricately designed to remove as much of this distortion based error as possible. If the simple standard isoparametric formulation shown above is used, the trapaziodal elements (mesh “e” above) would actually “lock” (become very stiff) and the errors in the displacements would be huge (over 90%). For this reason, it is critical that sophisticated, well-tested finite element codes be used for any critical analysis. Even then, it is wise to inspect meshes for regions where elements are highly distorted and attempt to create a less distorted mesh in that area.

V. STEP BY STEP INSTRUCTIONS FOR MODELING THE CANTILEVERED BEAM USING MSC/PATRAN

Preliminaries for using PATRAN include:

- a) Log on to the computer
- b) Click START (lower left corner of the Windows Desktop), go to Programs, Select MSC (common), Select MSC Patran9.0.

The instructions below give details for modeling the beam problem discussed above. Specifically, the 6 rectangular elements (mesh “b” above) is constructed. If one wishes to create any of the other meshes, the mesh creation section must be adapted to fit that mesh.

In the instructions below, the following abbreviations and terms will be used:

TM = Top Menu. This refers to the horizontal menu options residing at the top of the screen after PATRAN has been initiated.

RM = Right Menu. This refers to the menus that pop up after an option has been chosen from the top menu. These menus reside on the far right side of the PATRAN desktop.

SM = Subordinate Menu. This refers to the menus that pop up from options selected in the right menu.

Click = Unless otherwise stated, this indicates a click with the left mouse button.

Boldface will indicate text that occurs in the PATRAN menus.

Italics text will indicate text that you must enter into text boxes in the PATRAN menus or text that you choose in a menu scroll box.

1. Our first step is to create a new database:

From the TM choose **File**

In the resulting pull down menu choose **New**

A SM called **New Database** pops up

Turn on (checked) **Modify Preferences**

Under **File Name** enter *beam.db*

Click **OK**

2. Next set the analysis preference:

A **New Model Preferences** window will appear as a RM

Under **Tolerance** choose *Based on Model*

Set **Model Dimension** to *6.0*

Under **Analysis Code** choose *MSC/NASTRAN*

Choose **Analysis Type** = *Structural*

click **OK**

3. The geometry of the beam will be determined next:

From the TM choose **Geometry**

A RM called **Geometry** will result

Set **Action** = *Create*

Object = *Point*

Method = *XYZ*

Set the **Point ID** list to *1*

Set **Reference Coordinate Frame** to *Coord 0*

Turn off the **Auto Execute** button

Enter the following into the **Point Coordinates** list:

[0,0,0]

(note that PATRAN will accept either commas or blanks as separators between coordinates)

Click **Apply**

A point will appear in the main viewport at coordinates [0,0,0]

Back at the top of the RM called **Geometry**

Set **Action** = *Create*

Object = *Point*

Method = *XYZ*

Set the **Point ID** list to 2
Set **Reference Coordinate Frame** to *Coord 0*
Turn off the **Auto Execute** button
Enter the following into the **Point Coordinates** list:
[6,0,0]
(note that PATRAN will accept either commas or blanks as separators
between coordinates)

Click **Apply**

A point will appear in the main viewport at coordinates [6,0,0]

Back at the top of the RM called **Geometry**

Set **Action** = *Create*
Object = *Point*
Method = *XYZ*
Set the **Point ID** list to 3
Set **Reference Coordinate Frame** to *Coord 0*
Turn off the **Auto Execute** button
Enter the following into the **Point Coordinates** list:
[0,0.2,0]
(note that PATRAN will accept either commas or blanks as separators
between coordinates)

Click **Apply**

A point will appear in the main viewport at coordinates [0,0.2,0]

Back at the top of the RM called **Geometry**

Set **Action** = *Create*
Object = *Point*
Method = *XYZ*
Set the **Point ID** list to 4
Set **Reference Coordinate Frame** to *Coord 0*
Turn off the **Auto Execute** button
Enter the following into the **Point Coordinates** list:
[6,0.2,0]
(note that PATRAN will accept either commas or blanks as separators
between coordinates)

Click **Apply**

A point will appear in the main viewport at coordinates [6,0.2,0]

Back at the top of the RM called **Geometry**

Set **Action** = *Create*
Object = *Curve*
Method = *Point*
Set the **Curve ID** list to 1
Turn **Autoexecute** off
Set **Starting Point List** = *Point 1*
Set **Ending Point List** = *Point 2*
Click **Apply**

Back at the top of the RM called **Geometry**

Set **Action** = *Create*
Object = *Curve*
Method = *Point*
Set the **Curve ID** list to 2
Turn **Autoexecute** off
Set **Starting Point List** = *Point 3*
Set Ending Point List = *Point 4*
Click **Apply**

Back at the top of the RM called **Geometry**

Set **Action** = *Create*
Object = *Surface*
Method = *Curve*
Set the **Surface ID** list to 1
Set **Patran 2 Convention** *off*
Option = 2 *Curve*
Set **Manifold** *off* (not checked)
Set **Starting Curve List** = *Curve 1*
Set Ending Curve List = *Curve 2*
Click **Apply**

4. The boundary conditions are specified next:

From the TM choose **Load/BC's**

A RM called **Load/Boundary Conditions** will appear

Set **Action** = *Create*
Object = *Displacement*
Type = *Nodal*
Set **Current Load Case** = *Default*

Enter **New Set Name** as *l_cant*

(The name can be whatever name you wish. The name *l_cant* is chosen as this is for the cantilever of the left most nodes)

Click **Input Data...**

a SM called Input Data appears

Set **Load/BC Scale factor** = 1

Set **Translations** to <0,0,0>

Set **Rotations** to <0,0,0>

Be sure **Analysis Coordinate Frame** is *Coord0*

Click OK

(back in the **Load/Boundary Conditions** RM)

Click **Select Application Region**

A SM called **Select Application Region** appears

Turn on the **Geometry** (button down)

Click in box under **Select Geometric Entities**

In the Patran **Select Menu** (just to the left of the RM)

Click on the curve icon (just under the point icon)
In the main view port, select the left most vertical edge of the beam.

A **Selection Choices** SM appears

Choose *Surface 1.1*

(This will cause the words "Surface 1.1" to appear in the **Select Geometric Entities** box in the RM)

Click on **Add** just below this box

(This will remove the words "Surface 1.1 " from the **Select Geometric Entities** box and adds them to the **Application Region** box)

Click **OK**

(The **Load / Boundary Condition** RM appears again)

Click **Apply**

(3 displacement constraint arrows and 3 rotation constraint arrows should now appear on each point in the main viewport window on the extreme left edge of the beam. Numbers 1,2,3,4,5,6 will appear with the arrows to show that all 6 of the dof are constrained there)

5. The loads are specified next:

(Continuing on in the **Load/BC's** RM)

change **Action** = *Create*

Object = *Force*

Type = *Nodal*

Change the **New Set Name** to *r_point*

Click **Input Data...**

a SM appears

Enter the force vector $\langle 0, -1000, 0 \rangle$

leave the moments $\langle \quad \rangle$ (i.e. blank)

Click **OK**

(Continuing on in the **Load/BC's** RM)

Click **Select Application Region**

a small Patran select menu appears to the left edge of the RM

Click in this Patran select menu on the point icon

In the main viewport, click on the point 4 (top right corner of the beam)

A SM called **Selection Choices** menu appears.

Choose the **Point 4** option, not the **Curve** or **Surface** option)

(Point 4 will be added to the **Select Geometric Entities** list)

Click **Add**

(Point 4 will be added to the **Application Region** list)

Click **OK**

(**Load/BC's** menu now reappears)

Click **Apply**

(A vector with the 1000 unit downward load should appear on point 4 in the main viewport)

6. The finite element mesh is specified next:

From the TM choose **Elements**

A RM appears called **Elements**

Set **Action** = *Create*

Object = *Mesh*

Type = *Surface*

Set **Node Id** = *1*

Set **Element Id List** = *1*

Set **Global Edge Length** = *1.0* (This will create 6 elements. If you want to create only 2 elements (as is needed to answer question #1 below) then set the Global edge length to 3.0)

Set **Element Topology** = *Quad4*

Set **Mesher** = *Isomesh*

Click in the **Surface List** box

Click and drag to select the entire structure

The Words "Surface 1" should appear in the **Surface List**

Click **Apply**

Six elements will appear on the structure.

Set **Action** = *Equivalence*

Object = *All*

Type = *Tolerance Cube*

(The purpose here is to tie the nodes together that lie on top of one another)

Set the **Equivalencing Tolerance** to *.003*

Click **Apply**

(The command window at the bottom of the PATRAN desktop will tell you that 0 nodes were deleted. This step will become critical if, in more complicated models, you are attempting to join portions of a model which have been meshed separately.)

7. The materials are specified next:

On the TM select **Materials**

a RM will appear called **Materials**

Set **Action** = *Create*

Object = *Isotropic*

Method = *Manual Input*

Click **Material Name** box

Input the name to be *beam_mat1*

Click **Input Properties** box

SM called **Input Options** appears
Input **Elastic Modulus** = $200.0E9$
Input **Poisson** = 0.0
Click **OK**
Back in the **Materials RM**
Click **Apply**

8. The properties for each element are assigned next:

On the TM select **Properties**

a RM will appear called **Element Properties**

Set **Action** = *Create*

Dimension = $2d$

Type = *Shell*

Click **Property Set Name** box

Enter *beam_prop*

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "beam_matl" in the **Material Property Sets** box
at the bottom of the SM

(the words m:beam_matl will appear in the **Material Name** box at
the top of the SM)

Click in the **Thickness** box

Enter 0.1

Click **OK**

(Back in the **Element Properties RM**)

Click **Select Members** box

a Patran **Select menu** will appear on the left edge of the RM

Click on the icon which contains the surface or face icon

Move the cursor arrow to a point to the left and above the highest, left-
most point on the beam. Click and hold down the left mouse button. Drag
the cursor (while holding down the mouse button) to a point to the right of
and below the right-most bottom node. A "selection box" is formed while
you drag. Release the button.

(The words Surface 1 will appear in the **Select Members** box)

Click **Add**

(The words Surface 1 appears in the **Application Region** box)

Click **Apply** in the **Element Properties** menu

(beam_prop will be added to the **Existing Property Sets** box)

9. The analysis is to be done is specified next:

On the TM select **Analysis**

a RM will appear called **Analysis**

Set **Action** = *Analyze*

Object = *Entire Model*

Method = *Full Run*

Click Translation Parameters

In the SM that appears, set **Data Output** = *Op2 and Print*

Click **OK**

Back in the RM **Analysis**

Set **Solution Type** = *Linear Static* (button down)

Click **OK**

Click **Apply**

(The analysis will take a few seconds to run. A SM indicating that MSC/Nastran is working may appear)

10. A graphical representation of the deformation can be produced.

A graphical representation of the deformation provides an easy way to help determine if you have constructed your model correctly.

On the TM select **Analysis**

Set **Action** = *Read Output2*

Object = *Results Entities*

Method = *Translate*

Click **Select Results File**

A SM appears called **Select File**

Click the file **beam.op2**

(You may need to look in your home or root directory to find the file. If this file does not exist, then you have made a mistake in constructing your model. Go to Explorer (right-click on Start and choose Explore) and find the file beam.log and beam.f06. Open these files by double clicking on them and search for the word "error" to determine what your mistake is).

beam.op2 then appears in the **File Name** box

Click **OK**

(back in the **Analysis** menu)

Click **Apply**

On the TM select **Results**

A RM will appear called **Results**

Set Action = *Create*

Object = *Quick Plot*

In the Select Fringe Result **box** click *Displacements, translational*

In the Apply Fringe Result **box** click *Displacements, translational*

Set Quantity = *Y Component*

Click **Apply**

(This will create the deformed plot)

Note that stresses can also be plotted from the **Results** menu by specifying them in the **Select Fringe Result** section.

11. Next you will end your MSC/PATRAN session by saving your database and exiting.
On the TM select **File**

From the pull down menu select **Save**

On the TM select **File**

From the pull down menu select **Quit**

VI. EXERCISES:

For many of the exercises below, it may be helpful to use the Results Template included after the exercises.

- a) Compare the FEA results with the analytic results for the tip deflection and stresses using two elements and using 6 elements along the axis of the beam (see figures “a” – “e” above). This is a small h convergence test. Plot (% error) Vs (number of elements). Assume a linear function from your 2 data points. If this linear assumption is correct, what is the least number of elements you would need to get 1% error in the displacements? What is the least number of elements you would need to get 10% error in the stresses? (Note 1% error in displacements and 10% error in stresses are sometimes used for standard error goals. In addition, because stresses usually converge more slowly than displacements, these two errors often occur for approximately the same number of elements.)
- b) Return to your "MacNeal - Harder" beam model. Rerun the analysis using the same structural geometry, boundary conditions, loading conditions and material properties as you used previously. Model the structure with the 4 meshes specified as meshes b-e in the “Geometric Properties” section above. Compare both the displacement and stress solutions from the FEA with their analytic counterpart. Determine which type of distortion appears to be most detrimental to the FEA results by recording specific error percentages for all 4 meshes for both displacements and stresses.
- c) Noting that the elements we are using are "shell" elements (that is they have both membrane AND bending dof), rerun the four 6-node meshes. This time load the structure in the "out-of-plane" or Z direction. To do this you will need to apply 2 equal loads to the 2 nodes on the tip of the beam. These loads must be in the Z-direction. As the thickness is not the same as the width of the beam, your analytic answers for the tip deflection will be different than when you loaded the beam in the - Y direction. Again, compare the percent errors for both displacements and stresses for the meshes b-e.

- d) Determine the numerical values for the Jacobian matrix for an element in one of the distorted meshes.
- e) Run the 6-element distorted-mesh problems with the load parallel to the long axis (in axial loading) and discuss the resulting errors for both displacements and stresses.
- f) Rerun the analysis using only rectangular elements. Change the Poisson's ratio to 0.3 (normal Poisson's ratio for steel). Run the analysis using 2, then 6 then 480 elements (by setting the global edge length to 3, then 1 then 0.05 respectively). Record the displacement and stress results for the 3 meshes. Compute the percent errors for the stresses (using the analytic results as the baseline). Compare these errors with those found while using a Poisson's ratio of zero, Propose an explanation for the differences,
- g) Run the 6-element distorted-mesh problems with loads that create torsion and discuss the resulting errors for both displacements and stresses. Note that you will need to consult a Mechanics of Materials text for the analytic expressions for the displacements and stresses of a shaft with rectangular cross section.

++++
 RESULT TEMPLATE
 ++++

Analytic tip deflection for the membrane-based case = _____

Analytic tip deflection for the bending -based case = _____

Analytic max. stress for the membrane-based case = _____

Analytic max. stress for the bending -based case = _____

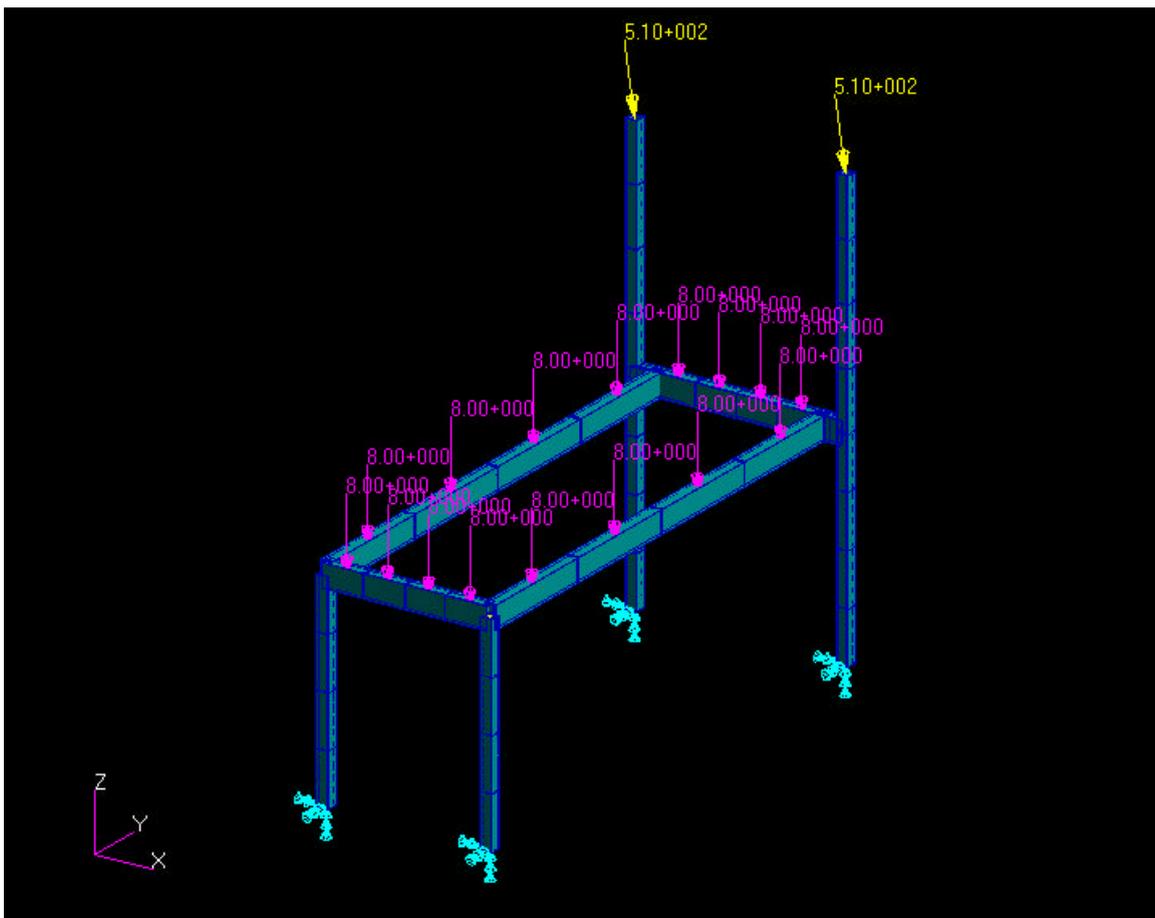
Loading Type: (bending with load in y, bending with load in Z, axial, torsion)

| MESH | TIP DISPL. | DISPL. ERROR | MAX. STRESS | STRESS ERROR |
|----------------------------|---------------|-----------------|----------------|-----------------|
| 2 element rectangular mesh | | | | |
| 6 element rectangular mesh | | | | |
| 10 degree parallelograms | | | | |
| 45 degree parallelograms | | | | |
| 45 degree trapezoids | | | | |

MSC/PATRAN TUTORIAL# 4
MODELING A FRAME STRUCTURE (WEIGHT BENCH)
USING BEAM ELEMENTS

I. THE PHYSICAL PROBLEM

The frame structure (weight lifting bench) below has the 4 legs that are attached to the floor. The weight of a user is assumed to be distributed across the rectangular box which sits in the horizontal plane. The weight of this user and accompanying weights is accounted for by a 8 lb per inch load distributed across the 108 inches of the horizontal rectangle. The weight on the uprights is assumed to be 500 lb max on each upright. This accounts for some impact load as well as the static force of a fully loaded bar. In addition to the vertical force, there is a 100 lb per upright force in the horizontal direction. This is intended to model the physics of someone pushing the bar horizontally (in the Y direction) against the cradle supports as they remove the bar to begin the bench press exercise.



II. THINKING ABOUT THE MECHANICS

The analytic solution for stresses and displacements for this problem is readily available if we think about the problem in sections. Any Mechanics of Materials text will provide equations for the stress and the displacements for built in and simply supported beams as well as axial loads. These results can be used to give basic analytic comparison solutions for certain sections of the structure.

III. GEOMETRIC AND FINITE ELEMENT MODEL

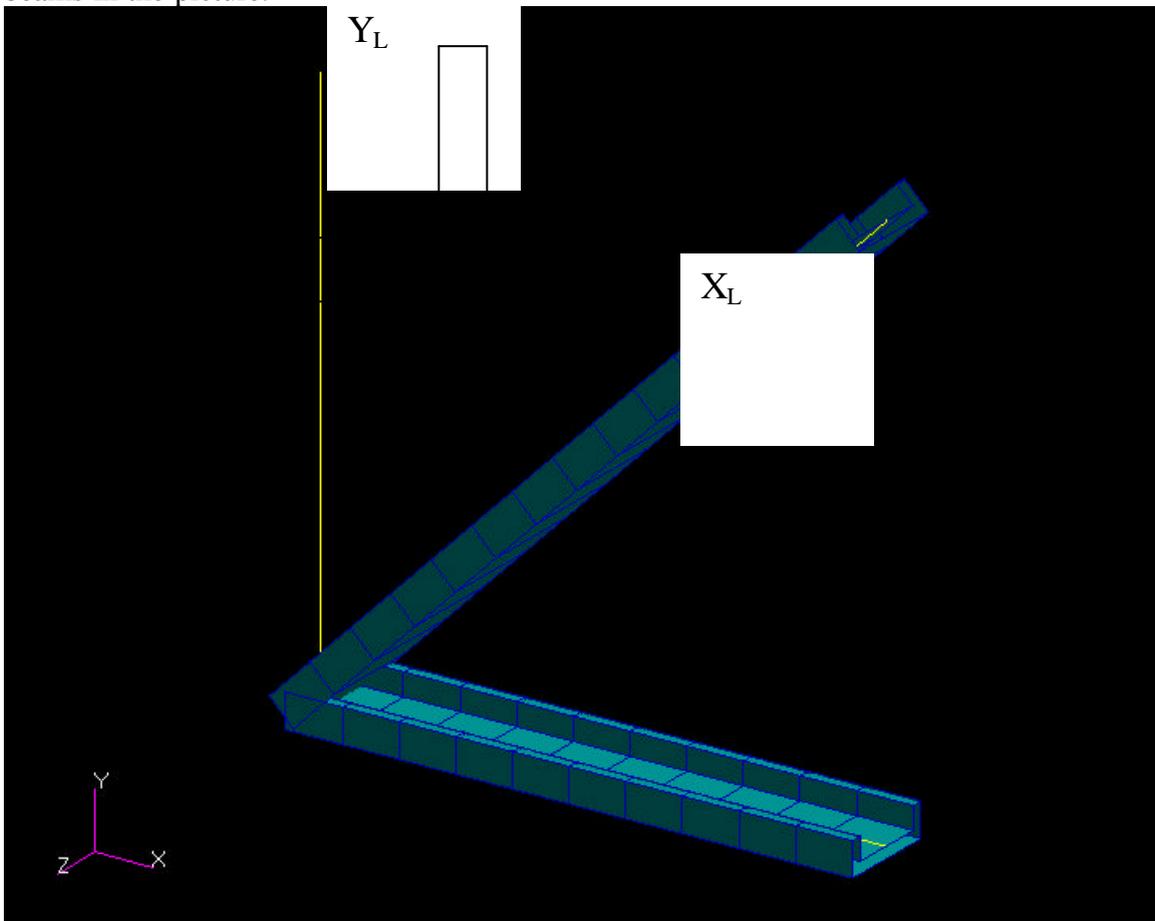
As is the standard procedure for building MSC/Patran models, we will build the geometry first and then construct a finite element mesh on that geometry. The geometry will proceed from creation of points to curves for this simple model. Next, we will use 2 node beam elements to model the frame. Next, the material and element properties will be entered. We will constrain the 3 displacement and 3 rotational degrees of freedom on the 4 legs. This creates the cantilevered or built-in, end conditions for these sections of the frame. Then we will place a point load of magnitude 500 in the $-Y$ direction on the top nodes of each of the uprights (where the weight bar would rest). A 100 lb load in the horizontal direction is also placed at that same node. A vertical load of 8 lb per inch is placed on the horizontal bench section (rectangle in the XY plane). Finally, the nodes must be equivalenced before the analysis is ready to run.

As can be seen in the step by step instructions below, Patran has a library of beam cross sections that can be used for frame analysis. These properties include various cross sections and wall thickness. One particular feature of note is the manner in which the orientation of the cross section is specified. The menu that allows you to pick the properties of the beam cross section requires a value for “Beam Orientation”. This value determines how the cross section will oriented. In particular, imagine that the graphic of the cross section (which is shown on the library menu) has a local coordinate system with X_L being the horizontal and Y_L being the vertical coordinates respectively (see figure below). Obviously, this means that Z_L is the coordinate down the long axis of the beam. If we label the “Beam Orientation” vector $\{bo\}$, then the following relationship can be used to specify our values for the components of $\{bo\}$.

$$\{bo\} \times \{Z_L\} = \{X_L\}.$$

An Example is shown below:

Consider the following cross section and the orientation of the cross section on the 2 beams in the picture.



For the section of the beam that has its long axis down the global X axis, the Beam Orientation vector $\{bo\}$ is set to $\{0,1,0\}$. This results in the orientation of the cross section as shown because $\{bo\} \times \{Z_L\} = \{X_L\} \rightarrow \{0,1,0\} \times \{1,0,0\} = \{0,0,1\}$. So the choice of $\{bo\} = \{0,1,0\}$ results in the global Z axis (i.e. $\{0,0,1\}$) being the local X-axis as seen in the graphic of the cross section. Note that this same choice for $\{bo\}$ will result in the orientation for the section of the beam that has its long axis in the $\{1,1,0\}$ direction above. This is because, for that case $\{bo\} \times \{Z_L\} = \{X_L\}$. $= \{0,1,0\} \times \{1,1,0\} = \{0,0,1\}$. This procedure is used below, in the step-by-step procedure, to determine the choice of $\{bo\}$ in the beam library menu.

IV. FINITE ELEMENT THEORY

The exact details of the formulation of the 2 node beam elements in MSC/Nastran is rather complicated. However, the basic formulation of an isoparametric 2 node beam

element is not extremely difficult and will provide us with sufficient background information to begin to understand the general application areas and “h” convergence of these elements. This basic formulation for the 2 node isoparametric beam can be found in the almost any Finite Elements text (see for example Finite Elements for Stress Analysis, R.D. Cook, John Wiley & Sons, 1995) .

V. STEP BY STEP INSTRUCTIONS FOR MODELING THE FRAME USING MSC/PATRAN & MSC/NASTRAN

Preliminaries for using PATRAN include:

- a) Log on to the computer
- b) Click START (lower left corner of the Windows Desktop), go to Programs, Select MSC (common), Select MSC Patran9.0.

The instructions below give details for modeling the beam problem discussed above. The instructions are NOT as detailed as I have given in other problems as I expect that you have begun to get a feel for how to do certain tasks in Patran.

In the instructions below, the following abbreviations and terms will be used:

TM = Top Menu. This refers to the horizontal menu options residing at the top of the screen after PATRAN has been initiated.

RM = Right Menu. This refers to the menus that pop up after an option has been chosen from the top menu. These menus reside on the far right side of the PATRAN desktop.

SM = Subordinate Menu. This referees to the menus that pop up from options selected in the right menu.

Click = Unless otherwise stated, this indicates a click with the left mouse button.

Boldface will indicate text that occurs in the PATRAN menus.

Italics text will indicate text that you must enter into text boxes in the PATRAN menus or text that you choose in a menu scroll box.

1. Our first step is to create a new database:

From the TM choose **File**

In the resulting pull down menu choose **New**

A SM called **New Database** pops up

Turn on (checked) **Modify Preferences**

Under **File Name** enter *bench.db*

Click **OK**

2. Next set the analysis preference:

A New **Model Preferences** window will appear as a RM
 Under **Tolerance** choose *Based on Model*
 Set **Model Dimension** to 40.0
 Under **Analysis Code** choose *MSC/NASTRAN*
 Choose **Analysis Type** = *Structural*
 click **OK**

3. The geometry of the beam will be determined next:

From the TM choose **Geometry**

A RM called **Geometry** will result

Set **Action** = *Create*

Object = *Point*

Method = *XYZ*

Set the **Point ID** list to 1

Set **Reference Coordinate Frame** to *Coord 0*

Turn off the **Auto Execute** button

Enter the following into the **Point Coordinates** list:

[2,0,0]

(note that PATRAN will accept either commas or blanks as separators between coordinates)

Click **Apply**

A point will appear in the main viewport at coordinates [2,0,0]

Using the same approach, create each of the other points in this table

| Point | X | Y | Z |
|--------------|----------|----------|----------|
| 1 | 2 | 0 | 0 |
| 2 | 16 | 0 | 0 |
| 3 | 2 | 0 | 18 |
| 4 | 16 | 0 | 18 |
| 5 | 18 | 42 | 0 |
| 6 | 0 | 42 | 0 |
| 7 | 18 | 42 | 38 |
| 8 | 0 | 42 | 38 |
| 9 | 16 | 42 | 18 |
| 10 | 2 | 42 | 18 |
| 11 | 18 | 42 | 18 |
| 12 | 0 | 42 | 18 |

Back at the top of the RM called **Geometry**

Set **Action** = *Create*

Object = *Curve*

Method = *Point*

Set the **Curve ID** list to *1*

Turn **Autoexecute** off

Set **Starting Point List** = *Point 1*

Set Ending Point List = *Point 3*

Click **Apply**

Using the same approach, create each of the other curves in this table

| Curve | Beginning Point | Ending Point |
|--------------|------------------------|---------------------|
| 1 | 1 | 3 |
| 2 | 2 | 4 |
| 3 | 4 | 9 |
| 4 | 4 | 3 |
| 5 | 11 | 7 |
| 6 | 11 | 5 |
| 7 | 11 | 9 |
| 8 | 9 | 10 |
| 9 | 12 | 10 |
| 10 | 12 | 8 |
| 11 | 12 | 6 |
| 12 | 3 | 10 |

4. The boundary conditions are specified next:

From the TM choose **Load/BC's**

A RM called **Load/Boundary Conditions** will appear

Set **Action** = *Create*

Object = *Displacement*

Type = *Nodal*

Set **Current Load Case** = *Default*

Enter **New Set Name** as *cant*

(The name can be whatever name you wish. The name *cant* is chosen as this is for the cantilever of the leg ends which contact the floor)

Click **Input Data...**

a SM called Input Data appears

Set **Load/BC Scale factor** = *1*

Set **Translations** to *<0,0,0>*

Set **Rotations** to *<0,0,0>*

Be sure **Analysis Coordinate Frame** is *Coord0*

Click **OK**

(back in the **Load/Boundary Conditions** RM)

Click **Select Application Region**

A SM called **Select Application Region** appears

Turn on the **Geometry** (button down)

Click in box under **Select Geometric Entities**

A **Selection Choices** SM appears

Choose *points 1,2,5,6*

Click on **Add** just below this box

Click **OK**

(The **Load / Boundary Condition** RM appears again)

Click **Apply**

(3 displacement constraint arrows and 3 rotation constraint arrows should now appear on each point in the main viewport window on the extreme lower edge of the bench's legs. Numbers 1,2,3,4,5,6 will appear with the arrows to show that all 6 of the dof are constrained there)

5. The finite element mesh is specified next:

From the TM choose **Elements**

A RM appears called **Elements**

Set **Action** = *Create*

Object = *Mesh Seed*

Type = *Uniform*

Choose the **Number of Elements** option

Set the number of elements to *4*

We want each of the 12 curves to have 4 elements. To ensure this, in the

Curve List Box enter *Curve 1:12*

Click APPLY

A set of mesh seeds will appear to show the density of nodes.

Back at the top of the RM called **Elements**

Set **Action** = *Create*

Object = *Mesh*

Type = *Curve*

Set **Node Id** = *1*

Set **Element Id List** = *1*

Set **Global Edge Length** = *1.0*

Set **Element Topology** = *Bar 2*

Click in the **Curve List** box

Click and drag to select the entire structure

Click **Apply**

Four elements will appear on each of the curves in the structure.

Back at the top of the RM called **Elements**

Set **Action** = *Equivalence*

Object = *All*

Type = *Tolerance Cube*

(The purpose here is to tie the nodes together that lie on top of one another)

Set the **Equivalencing Tolerance** to *.02*

Click **Apply**

(The command window at the bottom of the PATRAN desktop will tell you that some nodes were deleted. This step is CRITICAL as it “attaches” the nodes together at the frame junctions)

6. The materials are specified next:

On the TM select **Materials**

a RM will appear called **Materials**

Set **Action** = *Create*

Object = *Isotropic*

Method = *Manual Input*

Click **Material Name** box

Input the name to be *beam_matl*

Click **Input Properties** box

SM called **Input Options** appears

Input **Elastic Modulus** *30.0E6*

Input **Poisson** = *0.3*

Click **OK**

Back in the **Materials** RM

Click **Apply**

7. The properties for each element are assigned next:

On the TM select **Properties**

a RM will appear called **Element Properties**

Set **Action** = *Create*

Dimension = *1d*

Type = *Beam*

Click **Property Set Name** box

Enter *square_prop*

We will now create the cross sectional properties for the parts of the bench that have square cross sections. These parts are the 4 legs and the 2 uprights(which would hold the actual weight bar)

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "beam_matl" in the **Material Property Sets** box at the bottom of the SM

(the words m:beam_matl will appear in the **Material Name** box at the top of the SM)

Just to the right of the **Section Name** box, set the option to **Dimensions**

Click in the **Section name** box and input *square_sect*

Just to the right of the **Bar Orientation** box, set the option to **Vector**

Click in the **Bar Orientation** box and enter the vector $\langle 1,0,0 \rangle$
Click on the **ICLBeam Library** button

A SM appears called **Beam Library**

Set **Action** = *Create*

Dimension = *Standard Shape*

Type = *Nastran Standard*

Set the **New Section Name** to *Square1*

Scroll through the various possible cross sections using the < and > buttons (under the 3x3 set of cross section pictures) until you find the hollow rectangular picture with constant wall thickness (on the lowest row). Click this graphic.

In the upper right part of the window:

Set **W** = *1.0*

Set **H** = *1.0*

Set **t1** = *.125*

Set **t2** = *.125*

If you want to see the information on the cross sectional properties (which will come in handy when doing the analytical comparison calculation later) click on the **Calculate/Display** button.

Click **Apply**

Click **OK** (if a menu asks if you wish to over write say **YES**)

Click **Cancel**

Back in the Input Properties Menu, click **OK**

Back in the Properties RM

Click in the **Select Members** box

Choose the 4 legs and the 2 uprights (this is 6 curves and all the vertical members)

Click **Add**

Click **Apply**

Now we'll create the properties for the horizontal members. These members will have a 2 in x 1 in hollow cross section with .125 wall thickness. In this case it is critical that the large dimension of the cross section be oriented to provide the max bending moment of inertia "I", so the larger (2 in) dimension must be the vertical dimension of the cross section.

Back in the RM called **Element Properties**

Set **Action** = *Create*

Dimension = *1d*

Type = *Beam*

Click **Property Set Name** box

Enter *rectY_prop*

We will now create the cross sectional properties for the parts of the bench that have rectangular cross sections and run in the Y direction

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "beam_mat1" in the **Material Property Sets** box at the bottom of the SM

(the words m:beam_mat1 will appear in the **Material Name** box at the top of the SM)

Just to the right of the **Section Name** box, set the option to

Dimensions

Click in the **Section name** box and input *rectY_sect*

Just to the right of the **Bar Orientation** box, set the option to

Vector

Click in the **Bar Orientation** box and enter the vector $\langle 1,0,0 \rangle$

Click on the **ICLBeam Library** button

A SM appears called **Beam Library**

Set **Action** = *Create*

Dimension = *Standard Shape*

Type = *Nastran Standard*

Set the **New Section Name** to *RectY*

Scroll through the various possible cross sections using the < and > buttons (under the 3x3 set of cross section pictures) until you find the hollow rectangular picture with constant wall thickness (on the lowest row). Click this graphic

Set **W** = 2.0

Set **H** = 1.0

Set **t1** = .125

Set **t2** = .125

If you want to see the information on the cross sectional properties (which will come in handy when doing the analytical comparison calculation later) click on the **Calculate/Display** button.

Click **Apply**

Click **OK** (if a menu asks if you wish to over write say

YES)

Click **Cancel**

Back in the Input Properties Menu, click **OK**

Back in the Properties RM

Click in the **Select Members** box

Choose the 2 horizontal members that have their long axis in the Y direction (curves 3 & 12)

Click **Add**

Click **Apply**

Back in the RM called **Element Properties**

Set **Action** = *Create*

Dimension = *1d*

Type = *Beam*

Click **Property Set Name** box

Enter *rectX_prop*

We will now create the cross sectional properties for the parts of the bench that have rectangular cross sections and run in the X direction

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "beam_matl" in the **Material Property Sets** box at the bottom of the SM

(the words m:beam_matl will appear in the **Material Name** box at the top of the SM)

Just to the right of the **Section Name** box, set the option to

Dimensions

Click in the **Section name** box and input *rectX_sect*

Just to the right of the **Bar Orientation** box, set the option to

Vector

Click in the **Bar Orientation** box and enter the vector $\langle 0,1,0 \rangle$

Click on the **ICLBeam Library** button

A SM appears called **Beam Library**

Dimension = *Standard Shape*

Type = *Nastran Standard*

Set the **New Section Name** to *RectX*

Scroll through the various possible cross sections using the < and > buttons (under the 3x3 set of cross section pictures) until you find the hollow rectangular picture with constant wall thickness (on the lowest row). Click this graphic

Set **W** = 2.0

Set **H** = 1.0

Set **t1** = .125

Set **t2** = .125

If you want to see the information on the cross sectional properties (which will come in handy when doing the analytical comparison calculation later) click on the **Calculate** button.

Click **Apply**

Click **OK** (if you are asked to overwrite, say **YES**)

Click **Cancel**

Back in the Input Properties Menu, click **OK**

Back in the Properties RM

Click in the **Select Members** box

Choose the 4 horizontal members that have their long axis in the X direction (curves 4,7,8,9)

Note that there are 4 members that have their long axis aligned with the X axis; not just 2. These 4 include 2 curves that attach the uprights to the rectangular horizontal supports.

Click **Add**

Click **Apply**

In order to see if the cross sections are correctly aligned, go to the TM = Display, then select Load/BC/Elem Props... in the RM that appears, Under Beam Display, change the default 1-D Line to 3-D Full Span and hit Apply (at the bottom of the SM). This will turn on display of the cross sections. If you wish to see the cross sections shaded, you can use the TM shading icon (solid shaded box, just to the right of the little wire frame icons)

8. The loads are specified next:

Click the **TM = Loads/BC**

The RM Loads/BC pops up. Set

Action = *Create*

Object = *Force*

Type = *Nodal*

Change the **New Set Name** to *weights*

Click **Input Data...**

a SM appears

Enter the force vector $\langle 0, 100, -500 \rangle$

Leave the moments $\langle \quad \rangle$ (i.e. blank)

Click **OK**

(Continuing on in the **Load/BC's** RM)

Click **Select Application Region**

a small Patran select menu appears close to the RM

Click in this Patran select menu on the point icon

In the main viewport, click on the points 7 & 8 (top of the uprights on the bench)

Add these points to the application region

Click **OK**

(**Load/BC's** menu now reappears)

Click **Apply**

(A vector with the 510 unit downward and backward load should appear on points 7 & 8 in the main viewport)

Back in the main RM Loads/BC

Action = *Create*

Object = *Distributed Load*

Type = *Element Uniform*

Set **NewSet Name** = **d-load**

Set Target Element Type = *1-d*

Click **Input Data**

In the resulting SM

Set the **forces** to $\langle 0, 0, 8 \rangle$

Leave the **moments** blank

Click **OK**

Back in the Loads/BC RM

Click **Select Application Region**

In the resulting RM

Turn on **FEM** as the **Geometry filter**

Select all the elements along the curves 3,4,8,12 (these are the 4 beams in the XY plane that form the rectangle)

Click **Add**

Click **OK**

Back in the RM click **Apply**

Note: if the forces that appear on the main view screen are not in the correct direction, then you probably flipped one of the curve beginning/ending points. The easiest way to fix this is to remove the distributed load from those elements where it is in the wrong directions and create a second distributed force set that has the values <0,0,-8> and apply it to these elements.

9. The analysis is to be done is specified next:

On the TM select **Analysis**

a RM will appear called **Analysis**

Set **Action** = *Analyze*

Object = *Entire Model*

Method = *Full Run*

Click Translation Parameters

In the SM that appears, set **Data Output** = *Op2 and Print*

Click **OK**

Back in the RM **Analysis**

Set **Solution Type** = *Linear Static* (button down)

Click **OK**

Click **Apply**

(The analysis will take a few seconds to run. A SM indicating that MSC/Nastran is working may appear)

10. A graphical representation of the deformation can be produced.

A graphical representation of the deformation provides an easy way to help determine if you have constructed your model correctly.

On the TM select **Analysis**

Set **Action** = *Read Output2*

Object = *Results Entities*

Method = *Translate*

Click **Select Results File**

A SM appears called **Select File**

Click the file **bench.op2**

(You may need to look in your home or root directory to find the file. If this file does not exist, then you have made a mistake in constructing your model. Go to Explorer (right-click on Start and choose Explore) and find the file bench.log and bench.f06. Open these files by double clicking on them and search for the word "error" or "fatal" to determine what your mistake is).

beam.op2 then appears in the **File Name** box

Click **OK**
(back in the **Analysis** menu)
Click **Apply**

Select the TM **Results**

A RM will appear called **Results**

Set Action = *Create*

Object = *Quick Plot*

In the Select Fringe Result **box** click *Displacements, translational*

In the Apply Displacement Result **box** click *Displacements, translational*

Set Quantity = *magnitude*

Click **Apply**

(This will create the deformed plot)

Note that stresses can also be plotted from the **Results** menu by specifying them in the **Select Fringe Result** section. You will want to use the VonMises stresses in this case as the X, Y or Z based stresses are, by default, in the local coordinate system for that beam and are not in the global (Coord 0) frame.

11. Next you will end your MSC/PATRAN session by saving your database and exiting.

On the TM select **File**

From the pull down menu select **Save**

On the TM select **File**

From the pull down menu select **Quit**

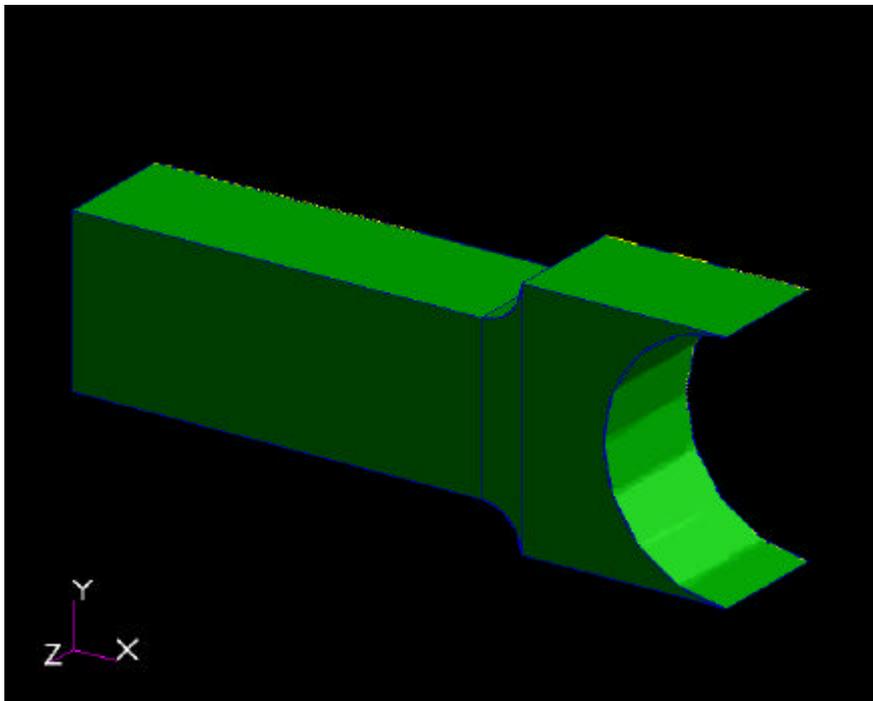
VI. EXERCISES:

1. Compare the FEA results with the analytic results for the mid-span deflection and stresses of a simply supported beam. To do this look at the mid-span deflection and stresses of either of the long horizontal members. Note that the BC of the ends of these members are Not really simply supported. There is some resistance to rotation of the cross section. However, neither is it truly a cantilevered BC. Therefore, if you calculate the midspan displacements sing simply supported BC, the analytic displacement will be an upper bound.
2. Create the Von Mises stress plot and the displacement plot. Do these make physical sense?
3. Compare the stresses in the uprights with the analytical P/A approximation using only the axial (Z component) part of the load. What do you conclude?
4. Compare the stresses in the uprights with the analytical My/I approximation using only the bending (y component) part of the load. What do you conclude?

MSC/PATRAN TUTORIAL # 5
MODELING A STABILIZATION FIXTURE WITH END PRESSURE
USING SOLID ELEMENTS

I. THE PHYSICAL PROBLEM

The structure below is designed to support a bearing on its right, curved edge. A similar part exists to hold the bearing on the other side. The left edge is cantilevered or "built in". This means that both the translations and the rotations are held to zero along this edge. A pressure load of magnitude 100 lb/in^2 in the negative X direction results from the bearing reaction. The material properties for the beam are $E = 10 \times 10^6 \text{ psi}$ (typical for aluminum) and $\nu = 0.3$. The part has a solid cross section with thickness in the Z-direction $t = 3 \text{ in}$.



II. THINKING ABOUT THE MECHANICS

The analytic solution for stresses and displacements for this problem is not readily available. However, any Mechanics of Materials text will provide equations for the max stress and the max displacement of simple problems that will provide upper or lower bounds for stresses and displacements. These analytic verifications will be discussed below.

Some basic questions to consider before creating the computational model are:

1. Where will the stresses be tensile and where will they be compressive?
2. What will be the magnitude and direction of the reaction forces/moments?

3. Where will the stresses be zero?
4. How do the displacements vary along the length (linear, quadratic etc.)?
5. What will the local effect of the pressure load be on the stresses?
6. Is the model fully constrained from rigid body rotations and displacements?

Answering these questions qualitatively, along with the quantitative analytical solutions for the max stress and displacement will provide reinforcement that your computational model is correctly constructed.

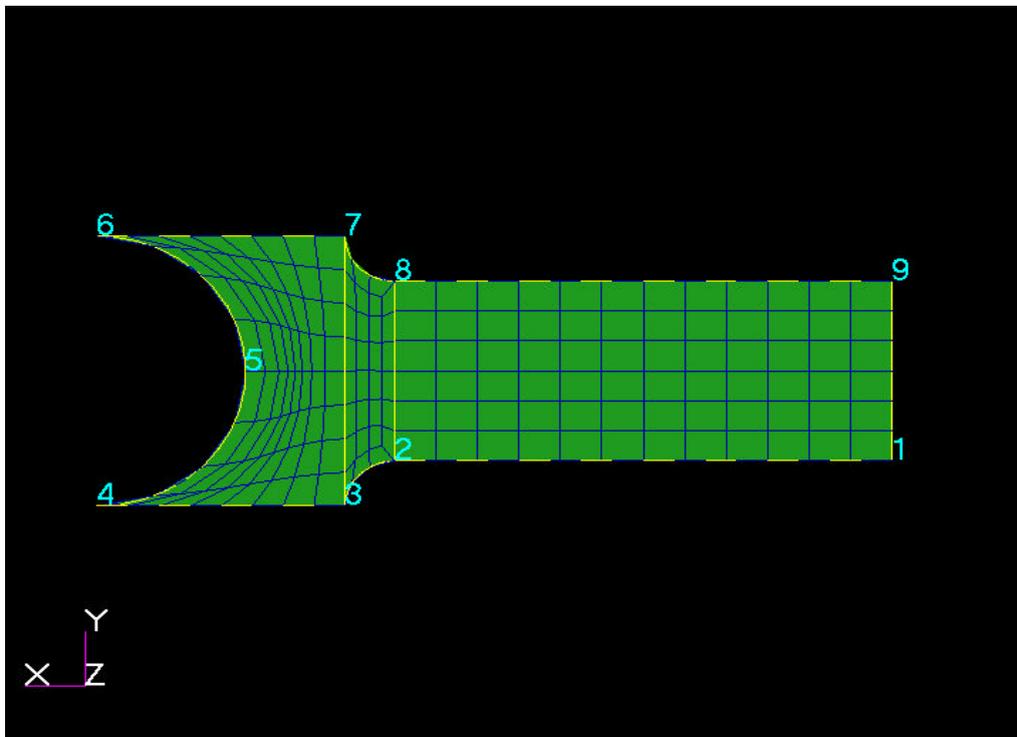
III. CREATING THE GEOMETRIC AND FINITE ELEMENT MODEL

1. Create the Geometry

Create the 3-d object below according to the following steps:

Create the points shown with coordinates as in the table

| Point | x-coord | y-coord | z-coord |
|-------|---------|---------|---------|
| 1 | 0 | 0 | 0 |
| 2 | 10 | 0 | 0 |
| 3 | 11 | -1 | 0 |
| 4 | 16 | -1 | 0 |
| 5 | 13 | 2 | 0 |
| 6 | 16 | 5 | 0 |
| 7 | 11 | 5 | 0 |
| 8 | 10 | 4 | 0 |
| 9 | 0 | 4 | 0 |

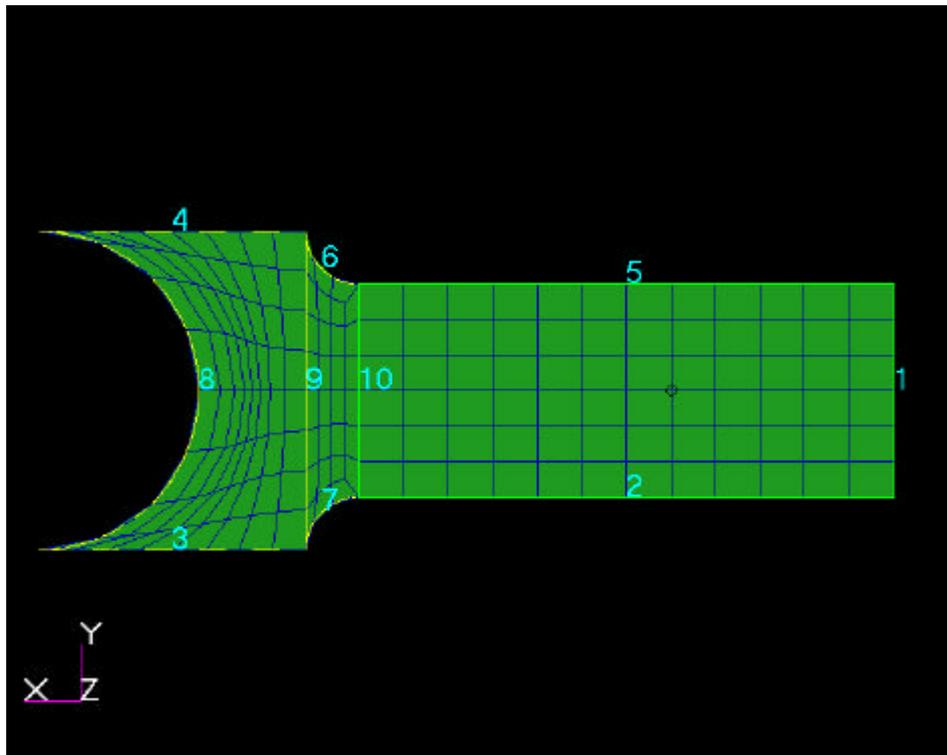


Next create curves between points 1 and 2, 3 and another curve between 4, 6 and another curve between 7, 8 and 9, 2 and another curve between 8, 3 and 7.

Create the curve between points 2 and 3 and between points 7 and 8 using the 2-d arc 2point using a radius of 1.

Create the arc between points 4, 5 and 6 using the 2d arc3point option.

Now create 3 surfaces. The instructions will use the curve numbers in the picture below. Please substitute the curve numbers from the curves you created.

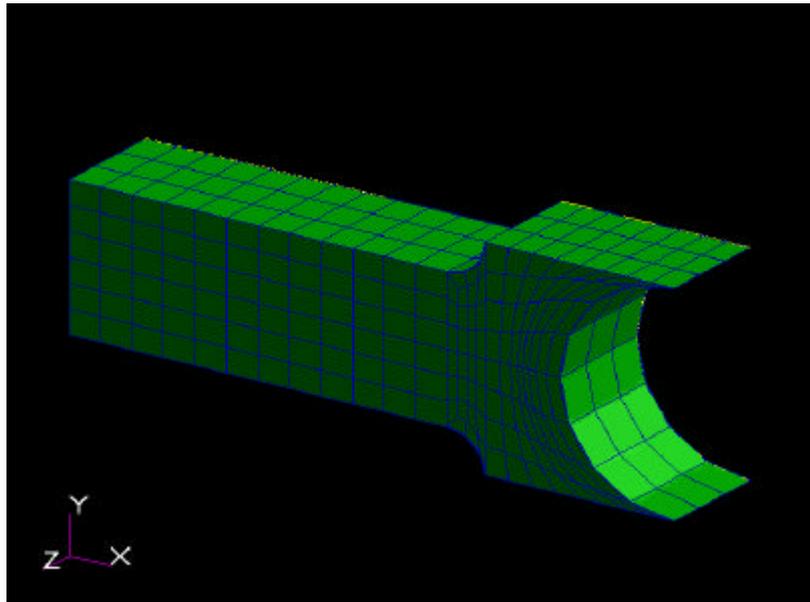


Create the first surface between the curves 1,2,5 and 10 using the Create/Surface/Edge command.

Create the 2nd surface between the curves 7, 10, 6 and 9 and the 3rd surface between the curves 3, 9, 4, and 8.

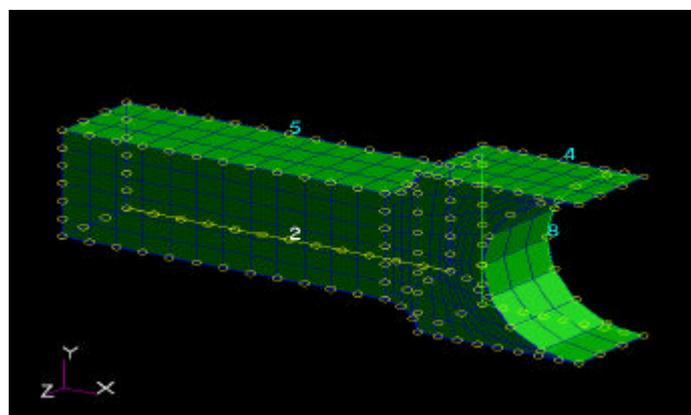
Now create solids of thickness 3 (in the Z direction) from each of the 3 surfaces using the Create/Solid/Extrude command. The translation vector will need to be <0,0,3>.

When you are done, the part will look like this.



2. Create the Finite Element Mesh

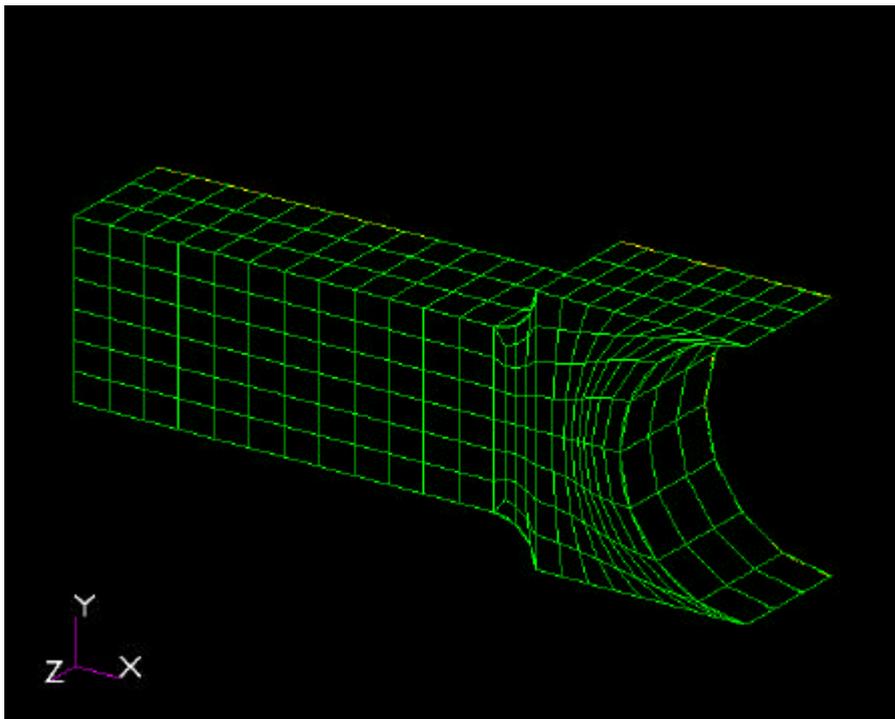
The first task is to seed the mesh. This is critical in this model as the model contains 3 separate solids which will need to be meshed separately. Then the nodes will need to be equivalenced so that the 3 solids are “attached” computationally. If this equivalencing is to work correctly, nodes along the interface between 2 solids will need to be coincident. The mesh seed will ensure this. Using the Create/Mesh Seed/Uniform option in the Element menu, create mesh seeds as shown below. Note that if your mesh seeds are a little different than the ones shown below, it will simply mean that you end up with a slightly different number and placement of elements. This should NOT affect the results of your analysis substantially except in one case. The distribution of elements in the Y direction must be constant across the part. For example if you have 6 elements across the far left edge, then you need to have 6 elements across the right curved edge and across other $X=\text{constant}$ planes in the part. The reason this is important is that if the element pattern is not symmetric in the in the Y direction, the part will experience a non-symmetric distribution of loads in the Y direction, resulting in non-physical displacements in the Y direction and also resulting in non-physical bending stresses.



Next place the actual mesh on the solids using the Create/Mesh/Solid. If the mesh seeds are done as shown, it will not matter what the global edge length is. Use the Hex8 elements. Use the isomesh mesh generator.

Next equivalence the nodes using the Equivalence/All/Tolerance cube command. The default tolerance is fine. This should indicate approximately 64 nodes were deleted. The graphics will show these nodes along the interface between the 3 solids.

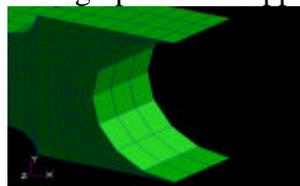
The completed mesh should look approximately as shown below.



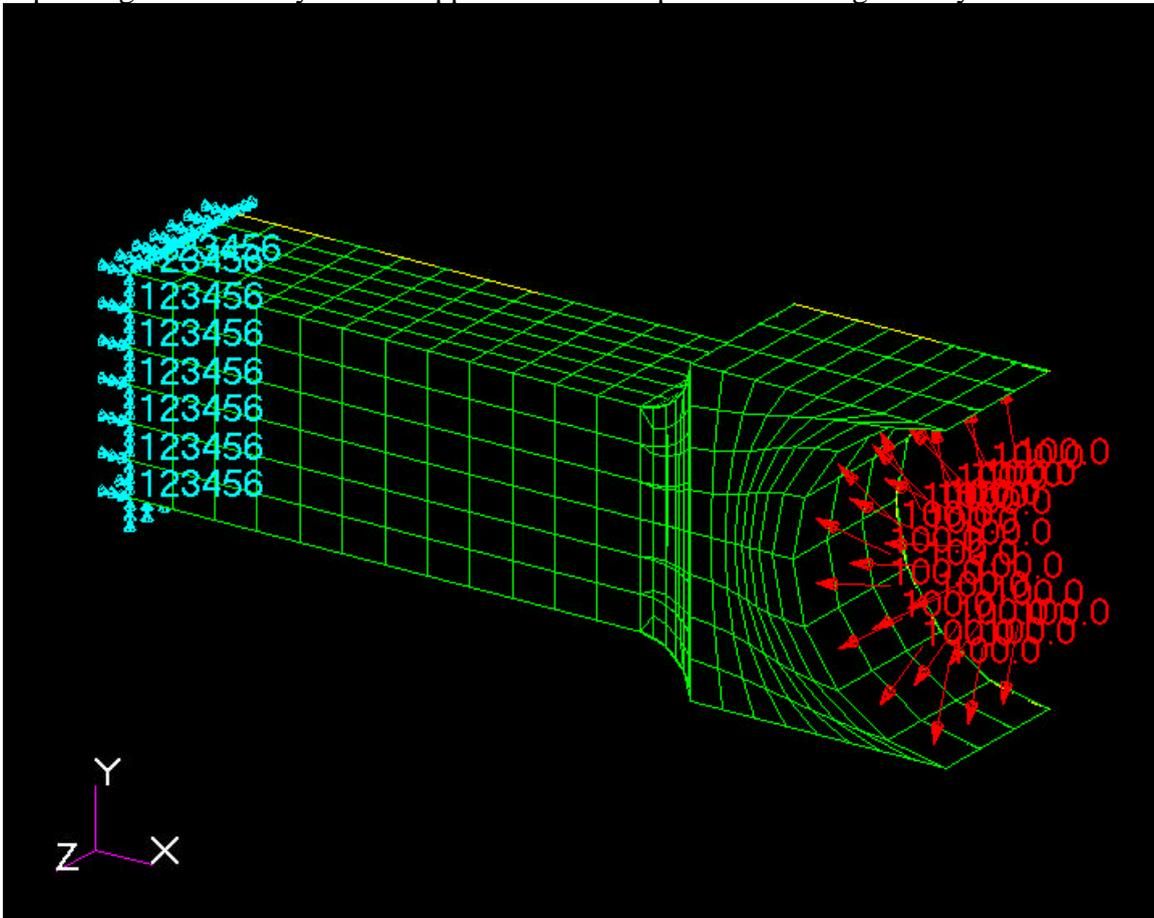
3. Create the BC and Loads

To clamp the left edge (edge away from the rounded bearing surface) use the Create/Displacement/Nodal command in the Loads/BC menu. Set all 3 displacements and all 3 rotations to zero and select apply to the left edge.

Next create the pressure load on the bearing surface by using the Create/Pressure/element Uniform command. Make sure the element target type is 3d. Choose a pressure of 100. To select the application surface (in the Select Application Surface) turn on the geometry button and then select the icon for “Face of a Solid” and choose the curved surface shown highlighted below. The BC and loads graphics will appear as shown.



The BC on the left and the pressure load on the right might look different on your part depending on whether you have applied the BC or pressure to the geometry .



4. Create the Material and Property

The material for the part is aluminum which has a E of $10e6$ psi and a Poisson's ratio of 0.3. Using the Materials menu create an isotropic homogeneous material with these properties.

The properties set is made using the command Create/3D/Solid in the properties menu. Input the properties simply as the material you just created. Select the entire part to have these properties.

5. Do the Analysis

In the analysis menu, use the command Analyze/Entire Model/Full Run . Set the Translation Parameters to output the *.op2 file. Read in the analysis results Using the command Read Output2/Result Entities/ Translate. Select the appropriate results (*.op2) file.

6. View the Results

In the results menu, use the command Create/Quick Plot. Make plots of the displacements ad the appropriate stresses.

VI. EXERCISES

1. Turn in the plots for the displacements and stresses. Do they make physical sense?

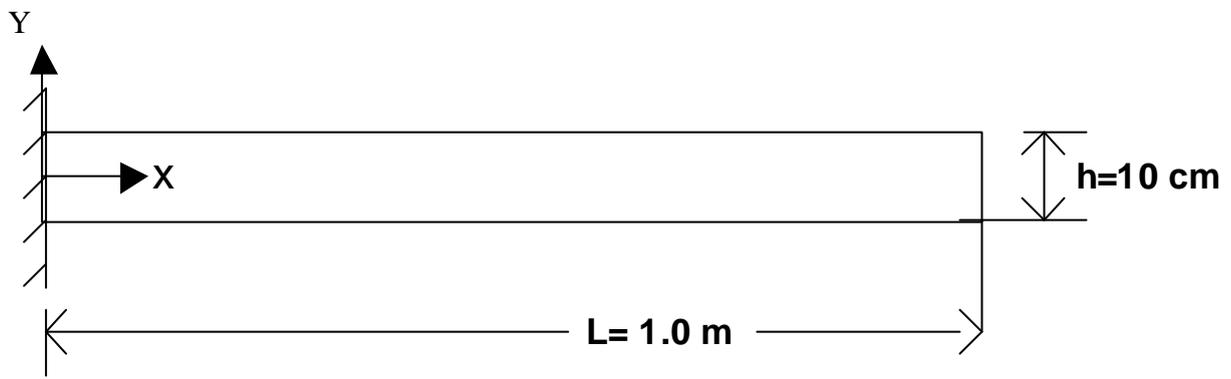
2. Find a way to get some substantial analytic verification for the model.

3. Look at the stress results and prescribe a weight savings measure that should not create stress related problems (i.e. determine where material can probably be removed). Computationally test your new design.

MSC/PATRAN TUTORIAL # 6
MODELING A CANTILEVERED BEAM'S VIBRATION
USING 4 NODE SHELL ELEMENTS

I. THE PHYSICAL PROBLEM

The beam below is cantilevered or "built in" on the left edge. This means that both the translations and the rotations are held to zero along this edge. The material properties for the beam are $E = 70 \times 10^9$ Pascals (typical for Aluminum) and $\nu = 0.3$. The beam has a solid rectangular cross section with thickness in the Z-direction $t = 0.01$ meters and height in the Y-direction $h = 0.1$ meters. We wish to find the mode shapes and associated vibration frequencies for this beam.



II. THINKING ABOUT THE MECHANICS

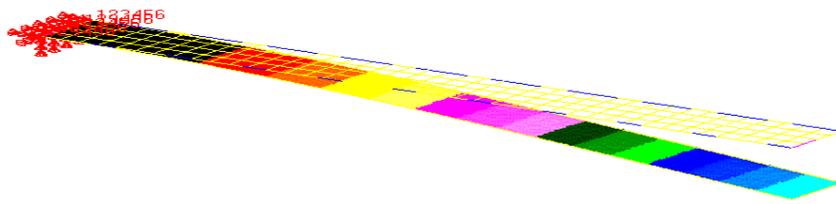
The analytic solution (modes shapes and natural frequencies) for this problem is readily available. Any vibrations text will provide equations for the mode shapes (eigenvectors) and the natural frequencies (eigenvalues). These equations are given below.

For the cantilevered beam with bending moment of inertia "I", Elastic (Young's) modulus "E", mass per unit length "m" and Length "L", the first 3 natural frequencies ω_{1-3} (rad/sec) are given by:

$$\omega_1 = 1.875^2 \sqrt{\frac{EI}{mL^4}} \quad \omega_2 = 4.694^2 \sqrt{\frac{EI}{mL^4}} \quad \omega_3 = 7.855^2 \sqrt{\frac{EI}{mL^4}}$$

Note that these correspond to the following 3 mode shapes which are all bending modes in the plane of the smallest value of "I".

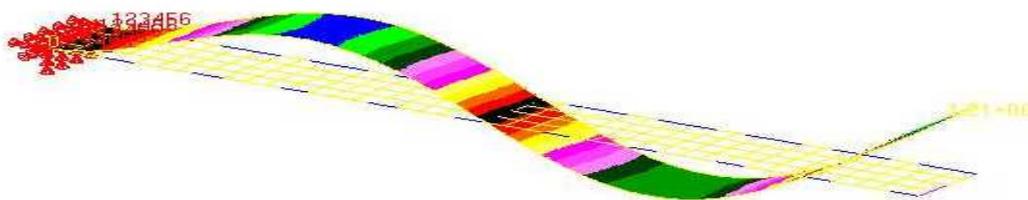
Modeshape 1:



Modeshape 2:



Modeshape 3



Some basic questions to consider before creating the computational model are:

1. Are there any other types of mode shapes that might occur (torsional, axial or bending in a different plane)?
2. What would be a reasonable frequency for the first mode shape?
3. Are there any constraint force checks that will help me validate the accuracy of my model?

Answering these questions qualitatively, along with the quantitative analytical solutions for the mode shapes and their associated natural frequencies will provide reinforcement that your computational model is correctly constructed.

III. GEOMETRIC AND FINITE ELEMENT MODEL

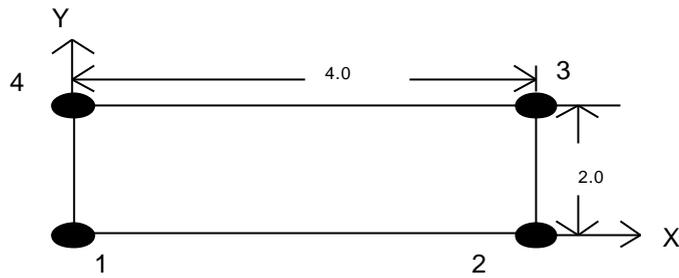
As is the standard procedure for building MSC/Patran models, we will build the geometry first and then construct a finite element mesh on that geometry. The geometry will proceed from creation of points to curves to surfaces for this simple model. Next, we will use 4 node shell elements to model the beam. Next, the material and element properties will be entered. We will constrain the 3 displacement and 3 rotational degrees of freedom on the left edge (for all nodes). This creates the cantilevered or built-in, end condition. Finally, the nodes must be equivalenced before the analysis is ready to run.

IV. FINITE ELEMENT THEORY

The exact details of the formulation of the 4 node shell elements in MSC/Nastran is rather complicated. However, the basic formulation of an isoparametric 4 node membrane element is not extremely difficult and will provide us with sufficient background information to begin to understand the vibration model studies. This basic form is constructed as follows:

Isoparametric Formulation of a 2-D Membrane Element [K] Matrix

Assume the element has the configuration shown below:



The physical and natural coordinate locations of the 4 nodes are:

| NODE | (x,y) | (\mathbf{x}, \mathbf{h}) |
|------|-------|------------------------------|
| 1 | (0,0) | (-1,-1) |
| 2 | (4,0) | (1,-1) |
| 3 | (4,2) | (1,1) |
| 4 | (0,2) | (-1,1) |

Our goal is to find the element stiffness matrix

$$\text{ASSUME: } [K] = \int_V [B]^T [E] [B] dV$$

ASSUME: 2 displacement degrees of freedom (dof) per node

With : $[B]$ = the strain - displacement matrix such that $[B]\{u\} = \{\mathbf{e}\}$

where: $\{u\}$ is the dof vector and $\{\mathbf{e}\}$ is the strain vector

$[E]$ = the constitutive matrix such that $[E]\{\mathbf{e}\} = \{\mathbf{s}\}$

where $\{\mathbf{s}\}$ is the stress vector and

V = volume.

Step 1: Interpolate the dof: $\begin{Bmatrix} u \\ v \end{Bmatrix} \approx [N] \{u_1, v_1, u_2, v_2, u_3, v_3, u_4, v_4\}^T = [N]\{u\}$

where $[N]$ is the shape function matrix $[N] = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix}$

and the rules for the shape functions are : 1) N_i must be =1 at node "i"

2) N_i must be =0 at any node not = "i"

This leads to the shape functions: $N_1 = \frac{1}{4}(1-\mathbf{x})(1-\mathbf{h})$; $N_2 = \frac{1}{4}(1+\mathbf{x})(1-\mathbf{h})$;

$N_3 = \frac{1}{4}(1+\mathbf{x})(1+\mathbf{h})$; $N_4 = \frac{1}{4}(1-\mathbf{x})(1+\mathbf{h})$

Step 2: Find the [B] matrix:

$$\text{Relevant strains are } \{\mathbf{e}\} = \begin{Bmatrix} \mathbf{e}_{xx} \\ \mathbf{e}_{yy} \\ \mathbf{g}_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} = [D] \begin{Bmatrix} u \\ v \end{Bmatrix}; \text{ but from step 1}$$

$$\begin{Bmatrix} u \\ v \end{Bmatrix} \approx [N] \{u\}$$

So $\{\mathbf{e}\} \approx [D][N]\{u\} = [B]\{u\}$ with $[B] = [D][N]$

Therefore,

$$[B] = \begin{bmatrix} N_{1,x} & 0 & N_{2,x} & 0 & N_{3,x} & 0 & N_{4,x} & 0 \\ 0 & N_{1,y} & 0 & N_{2,y} & 0 & N_{3,y} & 0 & N_{4,y} \\ N_{1,y} & N_{1,x} & N_{2,y} & N_{2,x} & N_{3,y} & N_{3,x} & N_{4,y} & N_{4,x} \end{bmatrix} \text{ where the commas denote}$$

partial differentiation.

Step 3: Use the Jacobian to find derivatives:

$$\text{Isoparametric Assumption: } \begin{Bmatrix} x \\ y \end{Bmatrix} = [N] \{x_1, y_1, x_2, y_2, x_3, y_3, x_4, y_4\}^T$$

i.e. the isoparametric assumption is that geometry can be interpolated using the same interpolation functions as the displacements.

$$\text{The Jacobian matrix } [J] = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \approx \begin{bmatrix} N_{1,x} & N_{2,x} & N_{3,x} & N_{4,x} \\ N_{1,h} & N_{2,h} & N_{3,h} & N_{4,h} \end{bmatrix} \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ x_3 & y_3 \\ x_4 & y_4 \end{bmatrix}$$

$$\text{and from chain rule } \begin{Bmatrix} N_{i,x} \\ N_{i,y} \end{Bmatrix} = \begin{bmatrix} \mathbf{x}_{,x} & \mathbf{h}_{,x} \\ \mathbf{x}_{,y} & \mathbf{h}_{,y} \end{bmatrix} \begin{Bmatrix} N_{i,x} \\ N_{i,h} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} N_{i,x} \\ N_{i,h} \end{Bmatrix}$$

So in this particular case: $[J] = \frac{1}{4} \begin{bmatrix} -1+h & 1-h & 1+h & -1-h \\ -1+x & -1-x & 1+x & 1-x \end{bmatrix} \begin{bmatrix} 0 & 0 \\ 4 & 0 \\ 4 & 2 \\ 0 & 2 \end{bmatrix} =$

$$\frac{1}{4} \begin{bmatrix} 8 & 0 \\ 0 & 4 \end{bmatrix} = \begin{bmatrix} 2 & 0 \\ 0 & 1 \end{bmatrix}$$

which implies that $[J]^{-1} = \begin{bmatrix} \frac{1}{2} & 0 \\ 0 & 1 \end{bmatrix}$

This allows us to find the entries in [B]

Step 4: Perform the numerical integration:

Assume that the element has constant thickness = t implies $[K] = t \int_A [B]^T [E] [B] dx dy$

Which, according to the rules of calculus can be written: $[K] = t \int [B]^T [E] [B] |J| dx dh$

where $|J|$ is the determinant of the Jacobian matrix.

Gaussian numerical integration is then used to find the final numbers for the element stiffness.

This takes the form: $[K] = h \sum_{j=1}^{ngj} \sum_{i=1}^{ngi} [B]^T [E] [B] |J| w_i w_j |_{(x_i, y_j)}$

Where ng_j and ng_i are the number of gaussian integration points in the “j” and “i” directions respectively and w_j and w_i are the associated gaussian weighting factors.

Understanding the Computational Vibration Analysis :

The elements as formed above must be assembled into a global stiffness matrix. In the same manner, element mass matrices are formed using the equation

$[M] = \rho \int [N]^T [N] |J| dx dh$. A similar form exists for the Rayleigh damping matrix

[C]. The stiffness, mass and damping matrices are then used in the dynamics equilibrium relationship $[M] \{\ddot{d}\} + [C] \{\dot{d}\} + [K] \{d\} = \{f\}$ where the over-dots indicated

derivatives with respect to time and {f} is the forcing function. This set of equations can be solved for the time history of the motion (transient dynamics) or for the eigenvalues and eigenvectors. For the vibration analysis, the damping and the forcing function are assumed to be zero. The resulting eigenvalue problem of the second kind is :

$[M] \{w\} + [K] \{d\} = \{0\}$ where eigenvalues are the natural frequencies **w** and the eigenvectors {d} give the node shapes.

V. STEP BY STEP INSTRUCTIONS FOR MODELING THE VIBRATION OF THE CANTILEVERED BEAM USING MSC/PATRAN

Preliminaries for using PATRAN include:

- a) Log on to the computer
- b) Click START (lower left corner of the Windows Desktop), go to Programs, Select MSC (common), Select MSC Patran9.0.

In the instructions below, the following abbreviations and terms will be used:

TM = Top Menu. This refers to the horizontal menu options residing at the top of the screen after PATRAN has been initiated.

RM = Right Menu. This refers to the menus that pop up after an option has been chosen from the top menu. These menus reside on the far right side of the PATRAN desktop.

SM = Subordinate Menu. This refers to the menus that pop up from options selected in the right menu.

Click = Unless otherwise stated, this indicates a click with the left mouse button.

Boldface will indicate text that occurs in the PATRAN menus.

Italics text will indicate text that you must enter into text boxes in the PATRAN menus or text that you choose in a menu scroll box.

1. Our first step is to create a new database:

From the TM choose **File**

In the resulting pull down menu choose **New**

A SM called **New Database** pops up

Turn on (checked) **Modify Preferences**

Under **File Name** enter *beam-vib.db*

Click **OK**

2. Next set the analysis preference:

A **New Model Preferences** window will appear as a RM

Under **Tolerance** choose *Based on Model*

Set **Model Dimension** to *10.0*

Under **Analysis Code** choose *MSC/NASTRAN*

Choose **Analysis Type** = *Structural*

click **OK**

3. The geometry of the beam will be determined next:

From the TM choose **Geometry**

A RM called **Geometry** will result

Set **Action** = *Create*

Object = *Point*

Method = *XYZ*

Set the **Point ID** list to *1*

Set **Reference Coordinate Frame** to *Coord 0*

Turn off the **Auto Execute** button

Enter the following into the **Point Coordinates** list:

[0,0,0]

(note that PATRAN will accept either commas or blanks as separators between coordinates)

Click **Apply**

A point will appear in the main viewport at coordinates *[0,0,0]*

Use this same procedure to create points at coordinates *[1,0,0]*, *[1,0.1,0]* and *[0,0.1,0]*

Back at the top of the RM called **Geometry**

Set **Action** = *Create*

Object = *Curve*

Method = *Point*

Set the **Curve ID** list to *1*

Turn **Autoexecute** off

Set **Starting Point List** = *Point 1*

Set Ending Point List = *Point 2*

Click **Apply**

Back at the top of the RM called **Geometry**

Set **Action** = *Create*

Object = *Curve*

Method = *Point*

Set the **Curve ID** list to *2*

Turn **Autoexecute** off

Set **Starting Point List** = *Point 3*

Set Ending Point List = *Point 4*

Click **Apply**

Back at the top of the RM called **Geometry**

Set **Action** = *Create*

Object = *Surface*

Method = *Curve*

Set the **Surface ID** list to *1*

Set **Patran 2 Convention** *off*

Option = *2 Curve*

Set **Manifold** *off* (not checked)

Set **Starting Curve List** = *Curve 1*

Set Ending Curve List = *Curve 2*

Click **Apply**

2. The finite element mesh is specified next:

From the TM choose **Elements**

A RM appears called **Elements**

Set **Action** = *Create*

Object = *Mesh*

Type = *Surface*

Set **Node Id** = *1*

Set **Element Id List** = *1*

Set **Global Edge Length** = *0.025*

Set **Element Topology** = *Quad4*

Set **Mesher** = *Isomesh*

Click in the **Surface List** box

Click and drag to select the entire structure

The Words "Surface 1" should appear in the **Surface List**

Click **Apply**

Set **Action** = *Equivalence*

Object = *All*

Type = *Tolerance Cube*

(The purpose here is to tie the nodes together that lie on top of one another)

Set the **Equivalencing Tolerance** to *.003*

Click **Apply**

(The command window at the bottom of the PATRAN desktop will tell you that 0 nodes were deleted. This step will become critical if, in more complicated models, you are attempting to join portions of a model which have been meshed separately.)

3. The boundary conditions are specified next:

From the TM choose **Load/BC's**

A RM called **Load/Boundary Conditions** will appear

Set **Action** = *Create*

Object = *Displacement*

Type = *Nodal*

Set **Current Load Case** = *Default*

Enter **New Set Name** as *l_cant*

(The name can be whatever name you wish. The name *l_cant* is chosen as this is for the cantilever of the left most nodes)

Click **Input Data...**

a SM called Input Data appears

Set **Load/BC Scale factor** = *1*

Set **Translations** to *<0,0,0>*

Set **Rotations** to *<0,0,0>*

Be sure **Analysis Coordinate Frame** is *Coord0*

Click **OK**

(back in the **Load/Boundary Conditions** RM)

Click **Select Application Region**

A SM called **Select Application Region** appears

Turn on the **FEM** (button down)

Click in box under **Select Nodes**

Use the cursor to highlight the set of nodes along the left vertical edge of the beam. There should be 5 nodes there.

Click **OK**

(The **Load / Boundary Condition** RM appears again)

Click **Apply**

(3 displacement constraint arrows and 3 rotation constraint arrows should now appear on each node in the main viewport window on the extreme left edge of the beam. Numbers 1,2,3,4,5,6 will appear with the arrows to show that all 6 of the dof are constrained there)

4. The materials are specified next:

On the TM select **Materials**

a RM will appear called **Materials**

Set **Action** = *Create*

Object = *Isotropic*

Method = *Manual Input*

Click **Material Name** box

Input the name to be *aluminum*

Click **Input Properties** box

SM called **Input Options** appears

Input **Elastic Modulus** = *70.0E9*

Input **Poisson** = *0.3*

Input the **Density** to be *2700*

Click **OK**

Back in the **Materials** RM

Click **Apply**

5. The properties for each element are assigned next:

On the TM select **Properties**

a RM will appear called **Element Properties**

Set **Action** = *Create*

Dimension = *2d*

Type = *Shell*

Click **Property Set Name** box

Enter *beam_prop*

Click **Input Properties**

a SM appears called **Input Properties**

Click in the **Material Name** box

Click on the word "aluminum" in the **Material Property Sets** box at the bottom of the SM

(the words m:aluminum will appear in the **Material Name** box at the top of the SM)

Click in the **Thickness** box

Enter *0.01*

Click **OK**

(Back in the **Element Properties** RM)

Click **Select Members** box

a Patran **Select menu** will appear on the left edge of the RM

Click on the icon which contains the surface or face icon

Move the cursor arrow to a point to the left and above the highest, left-most point on the beam. Click and hold down the left mouse button. Drag the cursor (while holding down the mouse button) to a point to the right of and below the right-most bottom node. A "selection box" is formed while you drag. Release the button.

(The words Surface 1 will appear in the **Select Members** box)

Click **Add**

(The words Surface 1 appears in the **Application Region** box)

Click **Apply** in the **Element Properties** menu

(beam_prop will be added to the **Existing Property Sets** box)

6. The analysis is to be done is specified next:

On the TM select **Analysis**

a RM will appear called **Analysis**

Set **Action** = *Analyze*

Object = *Entire Model*

Method = *Full Run*

Click Translation Parameters

In the SM that appears, set **Data Output** = *Op2 and Print*

Click **OK**

Back in the RM **Analysis**

Set **Solution Type** = *Normal Modes* (button down)

Click **OK**

Click **Apply**

(The analysis will take a few seconds to run. A SM indicating that MSC/Nastran is working may appear)

7. A graphical representation of the mode shapes can be produced.

A graphical representation of the mode shapes provides an easy way to begin to determine if you have constructed your model correctly.

On the TM select **Analysis**

Set **Action** = *Read Output2*

Object = *Results Entities*

Method = *Translate*

Click **Select Results File**

A SM appears called **Select File**

Click the file **beam-vib.op2**

(You may need to look in your home or root directory to find the file. If this file does not exist, then you have made a mistake in constructing your model. Go to Explorer (right-click on Start and choose Explore) and find the file beam- vib.log and beam.f06.

Open these files by double clicking on them and search for the word “error” to determine what your mistake is).

Beam-vib.op2 then appears in the **File Name** box

Click **OK**

(back in the **Analysis** menu)

Click **Apply**

On the TM select **Results**

A RM will appear called **Results**

Set Action = *Create*

Object = *Quick Plot*

In the Select Result Case **box** click **Default, Mode 1...**

In the Select Fringe Result **box** click *Eigenvectors, translational*

In the Apply Fringe Result **box** click *Eigenvectors, translational*

Set Quantity = *Magnitude*

Turn on the animation button (so it displays a check)

Click **Apply**

(This will create the animation of the first mode)

Investigate other, higher order mode shapes. Be sure to record data and screen captures needed to answer the questions below.

8. Next you will end your MSC/PATRAN session by saving your database and exiting.

On the TM select **File**

From the pull down menu select **Save**

On the TM select **File**

From the pull down menu select **Quit**

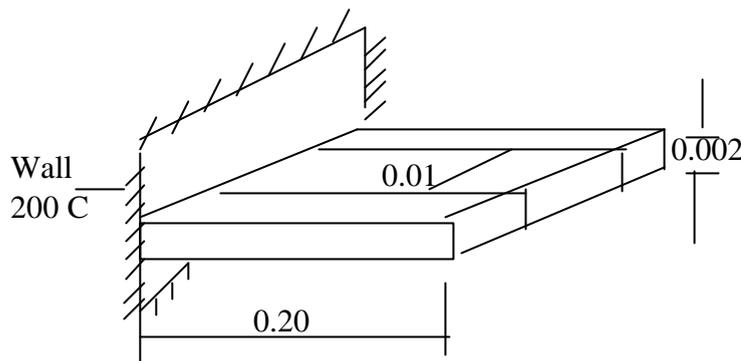
VI. EXERCISES:

- a) Compare the FEA results with the analytic results for the first 3 pairs of mode shapes and frequencies which are associated with bending of the beam in the direction of minimum “I”. You can use the analytic equations shown earlier to produce the analytic results.
- b) Study the first 5 mode shapes produced by the Nastran and comment on which modes are not associated with bending about the minimum “I” direction.
- c) Rerun the analysis using only .00625 as the global edge length (produces 4 times as many elements). Does a refinement in the mesh appear to produce more closely converged results?
- d) Change the Poisson’s ratio to 0.0. Rerun the analysis using the original global edge length of 0.025. Compare these errors with those found while using a Poisson’s ratio of 0.03, Propose an explanation for the differences.
- e) Identify the possible sources of that might make our results a poor model of the actual physical structure.

MSC/PATRAN TUTORIAL # 7
THERMAL ANALYSIS OF A COOLING FIN USING SHELL ELEMENTS

I. THE PHYSICAL PROBLEM

The problem you will model is a fin of aluminum alloy, 0.2 m long, 0.002 m thick and large width. This is the type of fin that might provide air-cooling on a motorcycle engine. For the finite element model, we consider a representative strip of the fin 0.01 m in depth (shown as the region between the dotted lines in the drawing). The 200-degree wall is representative of the hot temperature of the engine. Our goal is to find the temperature distribution down the fin. If the outside tip of the engine is too hot, it can be a safety concern. Heat is conducted down the fin (away from the heat source of the engine) and heat is also lost through convection from the top and bottom surfaces to the air. The ambient temperature of the air is known to be 25 C° and the convection coefficient (film coefficient) is known to be 30 (W/m²) . The fin itself is made of aluminum which has a conductivity of 177 (W/m² K).



II. THINKING ABOUT THE MECHANICS

The analytic solution for the temperatures for this problem is readily available. Any Heat Transfer text will provide equations for the temperature distribution of a fin considering conduction away from the heat source and convection from the top and bottom surfaces. These results can be used to give basic analytic comparison solutions for certain sections of the structure. Note that we assume no radiation occurs and that only the top and bottom surfaces have significant convection heat transfer (the convection from the edges of the fin is neglected). These assumptions are normal for a first level analysis where the temperatures are in the ranges used in this problem.

III. GEOMETRIC AND FINITE ELEMENT MODEL

As is the standard procedure for building MSC/Patran models, we will build the geometry first and then construct a finite element mesh on that geometry. The geometry will proceed from creation of curves to a surface for this simple model. Next, we will use 4 node 2-dimensional elements to model the fin. Next, the material and element properties will be entered. We will set the wall temperature and the convection characteristics for the top and bottom of the fin. Finally, the nodes must be equivalenced before the analysis is ready to run.

IV. FINITE ELEMENT THEORY

The exact details of the formulation of the 4 node 2-d elements in MSC/Nastran is rather complicated. However, the basic formulation of the 2-d thermal element is not extremely difficult and will provide us with sufficient background information to begin to understand the general application areas and convergence of these elements. This basic formulation for the 2-d thermal, linear, quasistatic element can be found in most any Finite Element Analysis text (see for example Finite Elements for Stress Analysis, by R.D. Cook, John Wiley & Sons, 1995.) .

V. INSTRUCTIONS FOR MODELING THE FIN USING MSC/PATRAN & MSC/NASTRAN

Preliminaries for using PATRAN include:

- a) Log on to the computer
- b) Click START (lower left corner of the Windows Desktop), go to Programs, Select MSC (common), Select MSC Patran9.0.

The instructions below give details for modeling the thermal fin problem discussed above. The instructions are NOT as detailed as have been given in other problems as it is expected that you have begun to get a feel for how to do certain tasks in Patran.

In the instructions below, the following abbreviations and terms will be used:

TM = Top Menu. This refers to the horizontal menu options residing at the top of the screen after PATRAN has been initiated.

RM = Right Menu. This refers to the menus that pop up after an option has been chosen from the top menu. These menus reside on the far right side of the PATRAN desktop.

SM = Subordinate Menu. This referees to the menus that pop up from options selected in the right menu.

Click = Unless otherwise stated, this indicates a click with the left mouse button.

Boldface will indicate text that occurs in the PATRAN menus.

Italics text will indicate text that you must enter into text boxes in the PATRAN menus or text that you choose in a menu scroll box.

1. Our first step is to create a new database:

From the TM choose **File**

In the resulting pull down menu choose **New**

A SM called **New Database** pops up

Turn on (checked) **Modify Preferences**

Under **File Name** enter *fin.db*

Click **OK**

2. Next set the analysis preference:

A **New Model Preferences** window will appear as a RM

Under **Tolerance** choose *Based on Model*

Set **Model Dimension** to *0.2*

Under **Analysis Code** choose *MSC/NASTRAN*

Choose **Analysis Type** = *Thermal*

click **OK**

3. The geometry of the beam will be determined next:

Select **Geometry** from TM.

On RM, select **Action**= Create, **Object**= Curve, **Method**= XYZ

Note **Curve ID List** has a 1.

Refer. Coordinate Frame should be Coord 0

Set **Vector Coordinates List** to 0.2 0 0 (You will be drawing lines (vectors) with these xyz components.)

Origin Coordinates List = 0 0 0 Click **APPLY**. (A line from origin to point 0.2,0,0 should appear on screen.)

Make second curve: With same vector, set **Origin Coordinates List** to 0 0.01 0.

Click **Apply**. (A second curve appears on the screen.)

Now create a surface between the curves.

On the **Geometry** RM, choose **Action**= Create; **Object**= Surface; **Method**= Curve.

Set **Option** to 2 Curve.

Note there is a **Starting Curve List** and **Ending Curve List**.

Click in the **Starting Curve List** box. .Select the first curve by using the mouse.

Click the small box on curve 1 on the screen. Click in the **Ending Curve List** box. Then click on curve 2. Note a surface is created.

4. Create the finite elements.

On the TM select **Elements** and get a RM.

Choose **Action**= Create; **Object**=Mesh; **Type**= Surface

Choose the size of the elements. Type in **Global Edge Length**: 0.01

Select **Isomesh**. Click in **Surface List** box. Select the Surface 1 with the cursor.

Click **Apply**. Note the model has 20 elements.

5. Create Boundary conditions

At TM, select **Load/BC's**. Get RM.

Now create the convection characteristics for the bottom of the fin

Chose **Action**=create, **Object**=convection, **type**=element uniform

Name the convection BC. In **New Set Name**, type *top-convec*

Select **Target Element Type** = 2D

Click **Input Data**. Get submenu. Type 30 for convection coefficient (w/m^2c) for top surface convection. Type 25 for ambient temp. Click **OK**.

Back in **Load/ BC** menu, click **Select Application Region** box

Select **FEM** as the Geometry Filter.

Click in **Select 2D Elements or Edges** box

Using mouse, click on all the elements. (Hold shift down for multiple selections.) Click **Add**. The application region box should list the elements 1:20. Click **OK**

Back in **Load/BC** menu click **Apply**.

Now create the convection characteristics for the bottom of the fin

Chose **Action**=create, **Object**=convection, **type**=element uniform

Name the convection BC. In **New Set Name**, type *bot-convec*

Select **Target Element Type** = 2D

Click **Input Data**. Get submenu. Type 30 for convection coefficient (w/m^2c) for bottom surface convection. Type 25 for ambient temp. Click **OK**.

Back in **Load/ BC** menu, click **Select Application Region** box

Select **FEM** as the Geometry Filter.

Click in **Select 2D Elements or Edges** box

Using mouse, click on all the elements. (Hold shift down for multiple selections.) Click **Add**. The application region box should list the elements 1:20. Click **OK**

Back in **Load/BC** menu click **Apply**.

Now create the base temperature BC. In **Load/BC** RM

Action=create, **Object**=Temp, **type**=nodal

In **New Set Name** type *Basetemp*. Click on **Input Data**.

In submenu **Input Data**, type 200 in **Temperature** box. Click **OK**.

Back in **Load/BC** SM, click on **Select Application Region**.

In submenu, select **FEM** as **Geometry Filter**.

Click on **Select Nodes**.

Using mouse, select the nodes 1 and 22 at the extreme left of the model. Click **Add**. Click **OK**.

Back in **Load/BC** menu, click **Apply**. (The screen should show 200 at nodes 1 and 22.)

6. Create and select material

On TM select **Materials**.

In submenu, **Action**=create, **Object**=isotropic, **method**=manual input.

In **Material Name** box, type *aluminum*. Click **Input Properties**.

In submenu, **Input Options**, enter thermal conductivity as 177 . ($\text{w/m}^2\text{k}$).

Click **OK**. If SM does not disappear, Click **Cancel**.

Back in the RM, Click **Apply**

In TM, select **Properties**

In submenu **Action=create**, **Object=2D**, **Type=shell**.

In **Property Set Name** type *shell_prop*.

Click on **Input Properties** In submenu, click on Aluminum in the Material Property Sets box.

M: Aluminum appears in the Material Name box at the top of the form.

Set **Thickness= 0.002** Click **OK**.

Back in **Element Properties**, click **Select members**.

Use mouse to select the entire model. (You can click and draw a box around the entire model to select it.) Click **Add**. Click **Apply**.

7. Load Boundary Conditions

In order to have both the convection on top and on the bottom as well as the and base temperature

BC on the model, all 3 boundary conditions must be combined into a single load case.

In TM, select **Load Cases**.

In SM, **Action=Create**, **Load case name**, type *fin_case*.

In **Description**, type: *h=30 on fin with base= 200C.*

and ambient=25C.

Click on **Assign/Prioritize BC** under the **Select Individual Loads/BC**

Click on **conve_bot-convec** and then on **conve_top-convec** and then on **temp_basetemp**

As you click on these each of the 3 is added to the **Assigned Load/BC**

At the bottom of the menu, click **OK**

Back in the Load Case RM, click **Apply**

8. Analyze (solve) for temperature.

In TM select **Analysis**. In SM, **Action= Analysis** **Object = entire model**,

Method = Full Run, **Job name = fin**

Click **translation Parameters** and set output to Op2 & Print

Click, OK

Back in the Analysis RM, click **Solution Type**

Choose **Steady State Analysis**

Click **OK**

Back in the Analysis RM, choose **Subcase Create**

Under **Available Subcases**, select *fin_case*

Under **Available Loadcases**, select *fin_case*

Click **Apply**

Click **Cancel**

Back in the Analysis RM, click **Subcase Select**

Under **Subcases for Solution Sequence 153**, select *fin_case*

Under **Subcases Selected**, click on Default (this removes default from the list)

Click **OK**

Back in Analysis SM Click **Apply**

9. To read in the results for post-processing

.In the RM=Analysis

Set **Action** = *Read Output2*; **Object** = *Results Entities*; **Method** = *Translate*

Click **Select Results File**

A SM appears called **Select File**

Click the file **fin.op2**

(You may need to look in your home or root directory to find the file. If this file does not exist, then you have made a mistake in constructing your model. Go to Explorer (right-click on Start and choose Explore) and find the file fin.log and fin .f06. Open these files by double clicking on them and search for the word “error” or “fatal” to determine what your mistake is).

fin.op2 then appears in the **File Name** box

Click **OK**

(back in the **Analysis** menu)

Click **Apply**

10. Select the TM Results

A RM will appear called **Results**

Set Action = *Create*

Object = *Quick Plot*

You can display different results, but the main focus will be on the temperatures.

11. Next you will end your MSC/PATRAN session by saving your database and exiting.

On the TM select **File**

From the pull down menu select **Save**

On the TM select **File**

From the pull down menu select **Quit**

VI. EXERCISES:

- I. Compare the FEA results with the analytic results for the problem found from a Heat Transfer text. How do the results compare. Discuss any discrepancies.
- II. Create the temperature plot. Does the distribution make physical sense? Why or why not?
- III. What assumptions are we making that might significantly affect the reliability of the results?