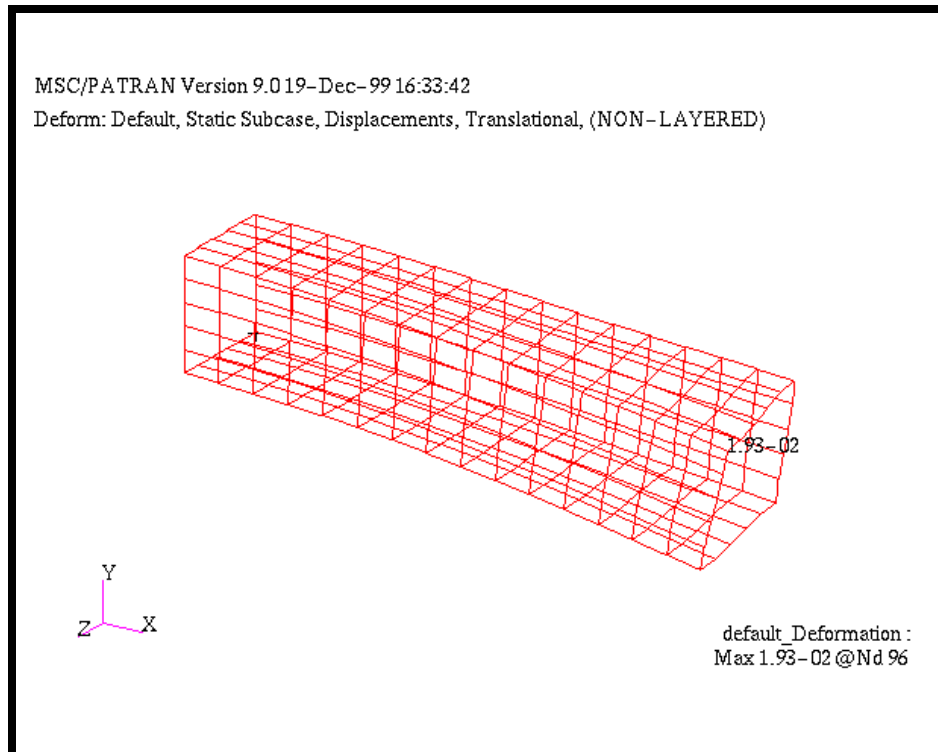

Exercise 1

Introduction to MSC.Patran



Objectives:

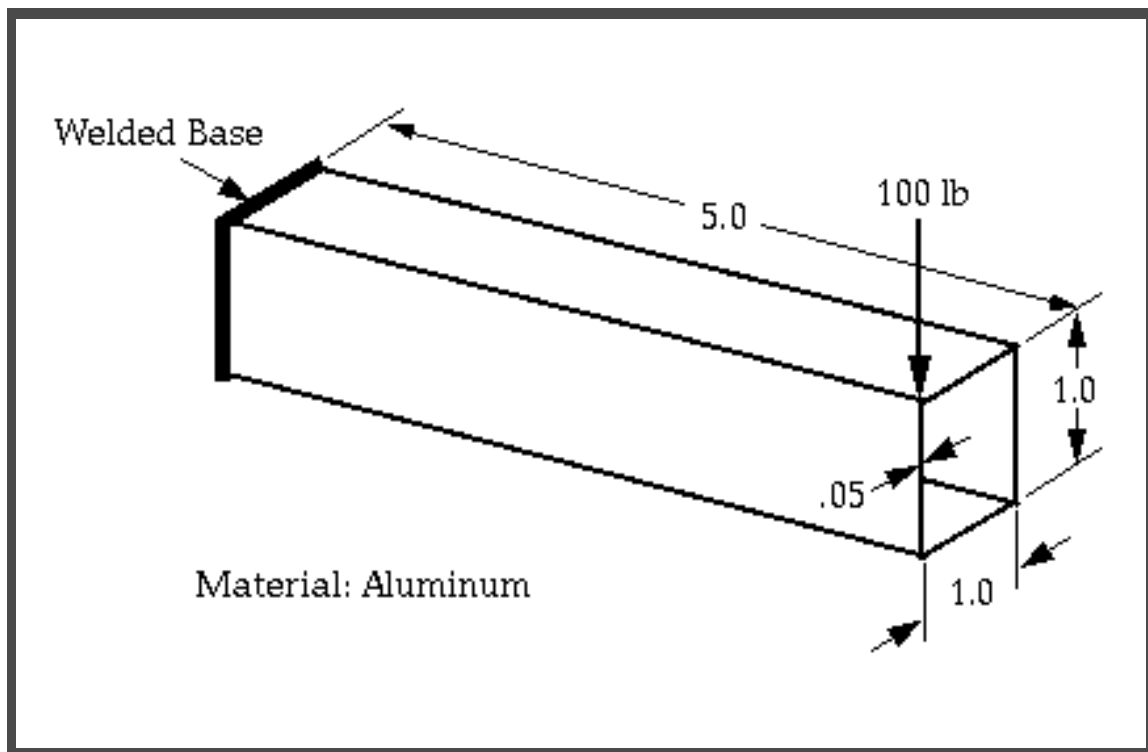
- Create geometry for a Beam.
- Add Loads and Boundary Conditions.
- Review analysis results.



Model Description:

This exercise will take you through the steps of modeling an open-ended aluminum box beam that is welded to a rigid surface. It has dimensions that are shown in the diagram below.

A downward 100-pound force will be applied, as shown. You will determine how much the box beam will deflect, as well as the maximum von Mises stress.



Suggested Exercise Steps:

- Create a new database and name it as **box_beam.db**.
- Create geometry.
- Apply boundary conditions and loads.
- Analyze model and view results.

Exercise Procedure:

*Note: In most MSC.Patran forms, the default setting for the **Auto Execute** button is on; thus, you do not need to press **Apply**.*

Open a New Database

1. Create a new database and name it **box_beam.db**.

File/New...

New Database Name

box_beam

OK

New Model Preferences

Tolerance:

◆ Default

Analysis Code:

MSC/PATRAN_FEA

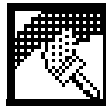
Analysis Type:

Structural

OK

Note: If the analysis code MSC/PATRAN_FEA is not available to you, choose another structural analysis code for which you have licenses.

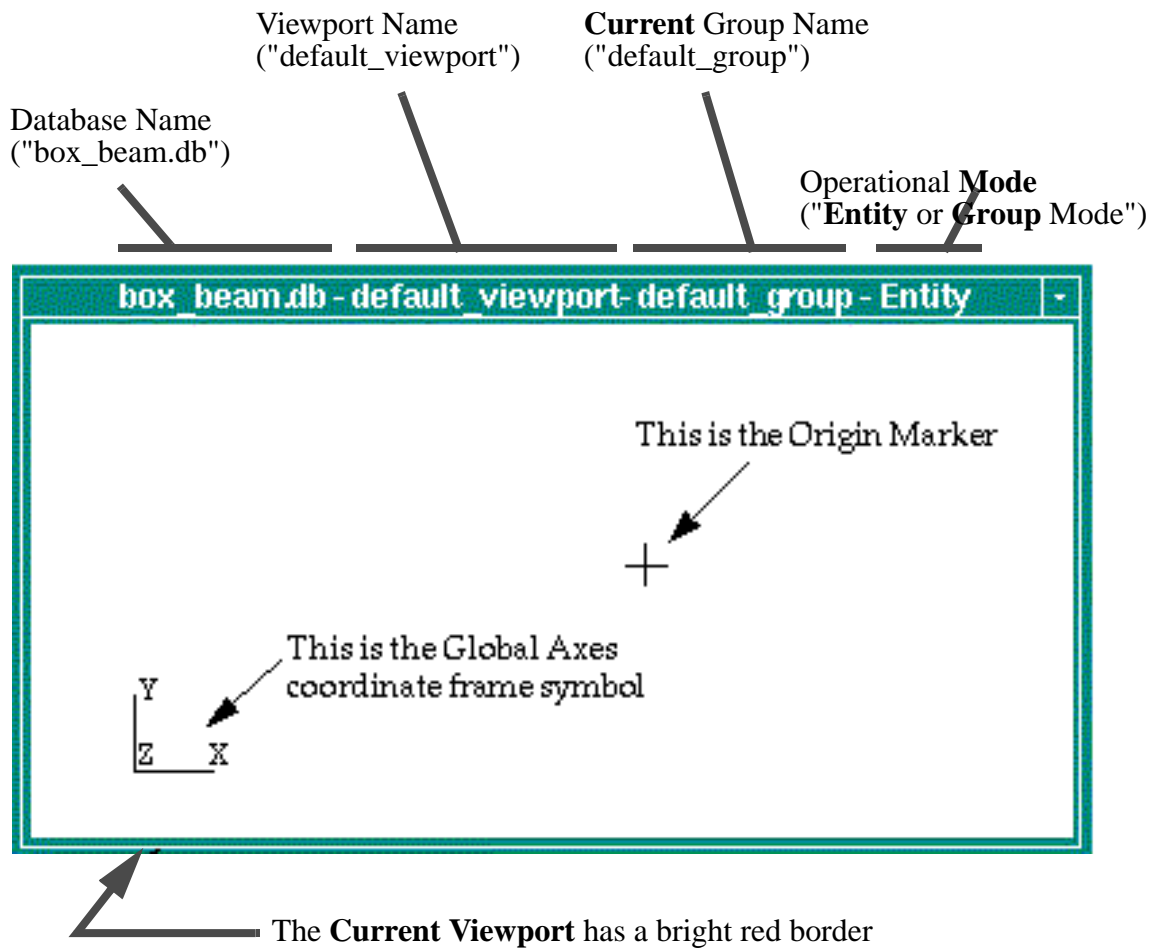
If the part of the screen which was covered by the New Model Preferences form is not redrawn, press the refresh button:



2. A Short Word on Viewports and Groups

After pressing **OK** on the *New Model Preferences* form, you should see a large graphics window, called the MSC.Patran **Viewport** (see below).

A viewport is a window in which you view your model. You can display more than one viewport, but to keep it simple we will be working with only one.



The top of the viewport lists the name of database you are working on, the name of the viewport, the name of the current group you are working with, and the mode of operation for displaying the model (Entity or Group mode).

A **Group** is a defined collection of geometry and/or finite element entities. Entities, such as a surface or an element, can belong to more than one group.

The **Current Group** is a specified group to which all newly created entities will belong. You can only specify one group at a time to be the current group for each viewport.

Create a Solid

3. Create a Solid

Though the box beam we will be creating is hollow, we will create a solid of the box beam and later, mesh the outside surfaces of the solid.

Keep in mind the dimensions of the beam are 5 inches long with a 1 inch cross section. We will create the solid with the long direction in the +X direction.

◆ Geometry

Action:

Create

Object:

Solid

Method:

XYZ

Vector Coordinates List

<5, 1, 1>

Origin Coordinates List

[0, 0, 0]

Apply

For fun, rotate the model by placing the cursor in the viewport and holding down the middle mouse button and dragging the mouse around. Notice that when you release the button a View/Fit View is automatically performed. You can switch from the rotate function to translate or zoom by entering Preferences/Mouse...

Change the Viewing Angle

4. Change the View Angle.

Instead of using the middle mouse button to change the view of the model, you can use the Viewing menu, by doing the following:

Viewing/Angles...

◆ Model Absolute

Angles

20 -10 0

Apply

Cancel

Change the Viewing Angle

5. Reset the Number of Display Lines

All surfaces and faces of solids are shown with additional lines, called **Display Lines**. These are not actual geometric boundaries, but they are **visualization** lines to help you see the interior curvature of the surfaces and faces.

Sometimes you may need to simplify the display of your model by lowering the number of Display Lines, and this is done as follows:

Display/Geometry

Number of Display Lines

1

Apply

Cancel

6. Create the Displacement Boundary Conditions.

MSC.Patran has this wonderful ability to associate the loads and boundary conditions (Loads/BCs) either with the geometry or with the finite element model.

The Loads/BCs associated with the geometry will be applied automatically to the nodes and elements when they are created. If you decide later to remesh the model, the Loads/BCs will be automatically revised. This means you can remesh your model as many times as you wish and you won't have to worry about reassigning the Loads/BCs to the mesh - MSC.Patran will take care of it for you!

For our box beam model, we want to "fix" the welded end of the beam by defining no movement in all six degrees of freedom, which is represented by six zeroes. (A null or blank value, instead of a zero, means the specific degree of freedom is free to move.)

To apply the fixed boundary condition:

Create Boundary Conditions

◆ Loads/BCs

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name

fixed

Input Data...

Translations <T1,T2,T3>

< 0 0 0 >

Rotations <R1 R2 R3>

< 0 0 0 >

Analysis Coordinate Frame

Coord 0

OK

Select Application Region...



Select Geometry Entities

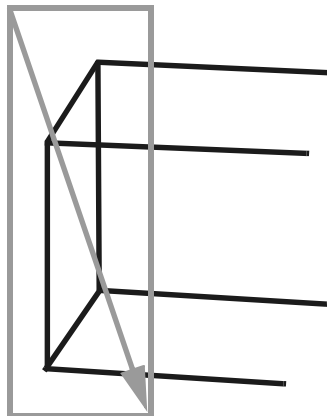


At the left of the menu you should see a select menu. By default the Geometric Entity icon is highlighted, which means MSC.Patran will look for all geometry entities when you cursor select entities in the viewport.

But we want to cursor select only the edges of the solid for our fixed boundary conditions. Thus, click on the Curve or Edge icon:



Cursor select the left end edges of our solid by using a rectangular cursor selection:



With the cursor at point “A”, **hold down** the left mouse button and **drag** the cursor to point “B”, so that you have drawn a rectangular box around the left end edges of the solid, as shown. Then, release the mouse button.

Note: MSC.Patran highlights the selected edges in red, and the “Select Geometry Entities” databox will list the solid edge IDs. (i.e., Solid 1.1.1, 1.1.2, 1.1.3, etc.).

If you are not happy with what you have selected, you can deselect the edges by using the **right** mouse button.

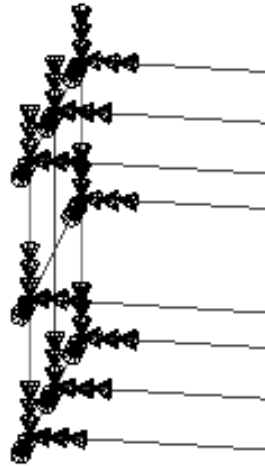
Finish creating the boundary condition by pressing the following menu buttons:

Add

OK

Apply

You should see light blue “cones” being drawn on the end of the solid at the Display Lines’ point locations (shown below). These represent the three fixed translational and the three fixed rotational boundary conditions. Remember, these have been applied to the geometry!



7. Apply the 100-pound Force Load.

Apply Load

Now we want to finish our Loads/BCs by applying our 100-pound downward force on the right corner of the solid. The force will be defined by a vector in the global rectangular coordinate frame, where we will have **zero pounds in the +X direction, 100 pounds in the -Y direction** and **zero pounds in the +Z direction**.

Again, we don't need to create the finite element mesh before applying our force. We can associated the force with the solid.

◆ Loads/BCs

Action:

Create

Object:

Force

Type:

Nodal

New Set Name

100_pounds_down

Input Data...

Force <F1 F2 F3>

< 0 -100 0 >

Analysis Coordinate Frame

Coord 0

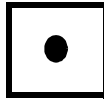
OK

Select Application Region...

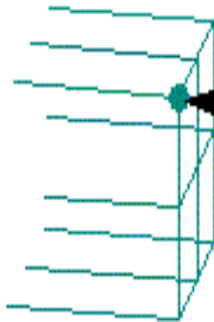
◆ Geometry

Select Geometry Entities

Since we want to apply our force to a point on the solid, switch the select menu icon to the **Point or Vertex icon**:



Cursor select the corner point of the right end of the solid, as shown:



Click on this corner of the solid with the left mouse button. MSC.Patran will highlight the point in red.

You should see “Point 7” entered in the “Select Geometry Entities” databox. If you cursor selected another point by mistake, press the right mouse button to deselect it.

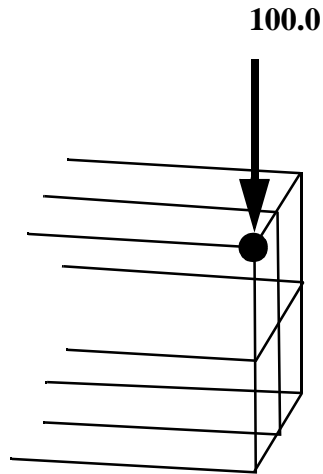
Finish creating the force by pressing the following menu buttons:

Add

OK

Apply

Now you should see a yellow arrow drawn on the selected corner point of the solid which is our 100-pound force:



8. Create a New Group

Before we go on and mesh our box beam model, we want to keep our soon-to-be finite element mesh in a group that is separate from our geometry model.

To do this, we will create a new, empty group called “fem_model” and make “fem_model” our current group.

Create a New Group

Group/Create...*New Group Name***fem_model**☒ **Make Current***Group Contents:***Add Entity Selection***Entity Selection***Apply****Cancel**

Note: “fem_model” has replaced “default_group” at the top of our viewport, which means “fem_model” is our new current group.

**Define
Mesh Seeds**

9. Define Mesh Seeds on one end of the Solid.

Before we mesh the outer surfaces of our solids, we want to take advantage of a feature called **Mesh Seeds**.

Mesh Seeds allow you to define exactly how many elements (and even what node spacing) you want on selected curves or edges of a surface or a solid.

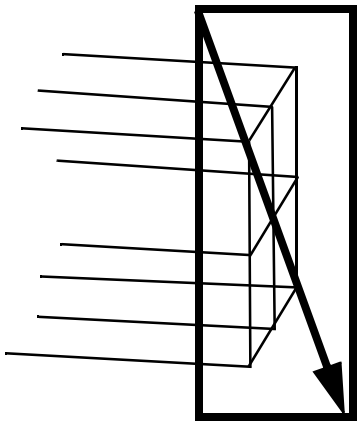
For our box beam model, we want to define mesh seeds of five elements in the Y and Z directions and 15 elements in the X direction. To do this, we do the following:.

◆ **Finite Elements**

Action:	Create
Object:	Mesh Seed
Type:	Uniform
◆ Number of Elements	
Number=	5
Curve List	

Because mesh seeds can only be associated with curves or with edges of surfaces or solids, the select menu only has one icon. This means MSC.Patran will only look for any curves or edges that you select.

Since we want to select the edges of the solid that are in the Y and Z directions, use the rectangular cursor select box and cursor select the edges on the **left end** or the **right end** of the solid (the illustration below shows the solid's right end edges being selected):



Use the rectangular cursor selection box and select the edges on the left or right end of the solid

Apply

Now we want to create our second mesh seed for 15 elements in the X direction:

◆ Number of Elements

Number=

15

Curve List

Select one of the 4 edges in the *middle* area of the solid. The selected edge of the solid will be **red**.

Apply

10. Mesh the Outside Faces of the Solid with Quad4s

Meshing

Now we're ready to create our finite element mesh. Though we have a solid, we want to create a surface mesh made of 4-noded quadrilateral elements (Quad4s). MSC.Patran allows you to create a surface mesh on the *faces* of our solid.

To do this, do the following:

◆ Finite Elements

Action:

Create

Object:

Mesh

Type:

Surface

Global Edge Length

0.1

Element Topology:

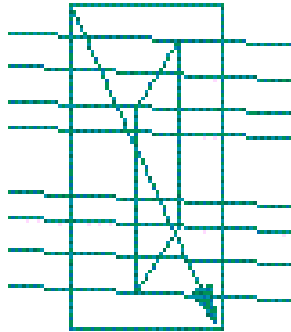
Quad4

◆ IsoMesh

Surface List

Notice the select menu will only allow you to select surfaces or solid faces, because the Type option on the Finite Elements form is set to "**Surface**". Again, either individually select the outside solid faces (not including the two ends) using shift/click, or:

you may now cursor select the faces in the *middle area* of the solid, using the rectangular cursor selection box:

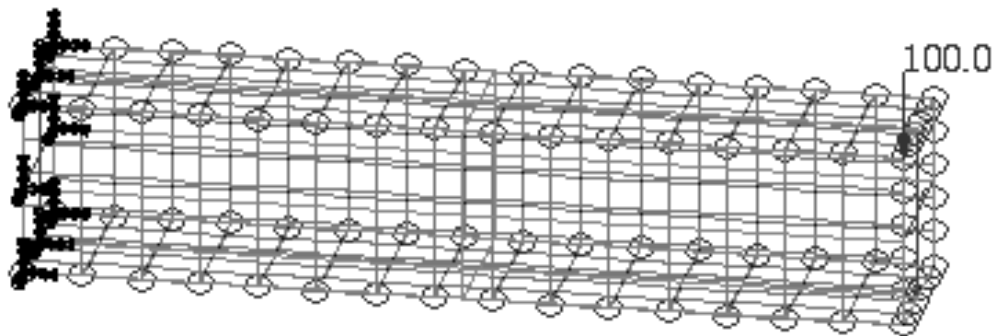


Use the rectangular cursor selection box and select the edges in the **middle** of the solid. Use Preferences/Picking and select Enclose any portion of entity. The selected edges of the solid will be red.

The Surface List databox should list "solid 1.3 1.4 1.5 1.6".

Apply

The model should now look like the following:



Equivalence

11. Equivalence the Coincident Nodes.

You may not realize it, but duplicate or coincident nodes were created along the edges between the neighboring solid faces. You will need to equivalence the nodes to remove each one of the two coincident nodes.

◆ Finite Elements

Action:

Equivalence

Object:

All

Type:

Tolerance Cube

Equivalencing Tolerance

0.005

Apply

Purple circles will appear which tell you where the coincident nodes were found and removed.

12. Align the Element Normals.

Element Normals

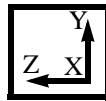
"Why bother?", you may ask? You may not realize it, but 2-D shell elements, like the Quad4s used in this box beam model, have a defined top and bottom surface for results output.

By default, the Quads4s we created have their positive surface normals (which are the elements' top surfaces) pointing in the positive global coordinate directions. But for postprocessing the results, we want the normals to be pointing outward from the box, regardless of their orientation in global XYZ space.

To show you what we mean, let's first plot the existing normal vectors by doing the following.

First, reset the view by looking down on the end of the box beam

Click on the Right Side View icon:



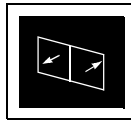
Now let's plot the element normal vectors:

◆ Finite Elements

*Action:***Verify***Object:***Element***Method:***Normals***Display Control*

◆ Draw Normal Vectors

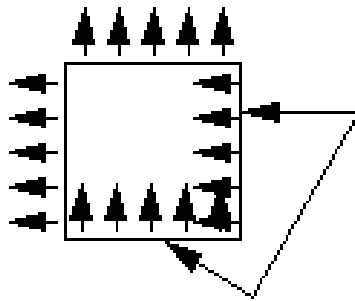
Make sure the Test Control icon looks like this:



It should state "Display Only" on the side

Apply

You should see red arrows being plotted that look like this:

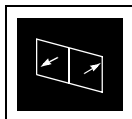


We want to reverse the element normal directions on these two sides of the box so they point outward.

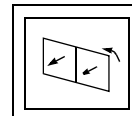
Notice the normals are aligned with the +Y and +Z global coordinate directions.

Now, let's reverse the normal directions of the two sides indicated above.

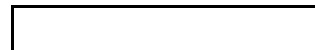
First, press the **Display Only** icon, under **Test Control**, so that it changes to the **Reverse Elements** icon:



Changes to:



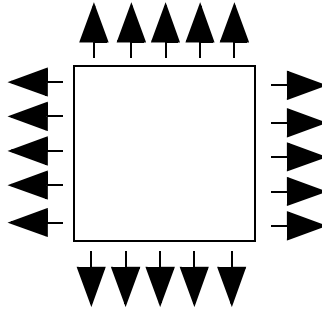
Guiding Element



Use the left mouse button and pick one element from the top surface which is pointing in the "correct" or outward direction.

Apply

You should see the element normals all pointing outward:



The normal vectors' red arrows can be erased by either pressing the Reset Graphics button, or by exiting the Finite Elements Application and continuing on to the next step.

13. Define the Material Properties.

Define Material Properties

Since the box beam is made of aluminum, we can assume the material is isotropic and it will behave in the linear elastic region. To create the material property:

◆ Materials

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name

aluminum

Input Properties...

Constitutive Model:

Linear Elastic

Elastic Modulus=

10.0e6

Poisson Ratio=

0.3

Density=

0.000259

Thermal Expand. Coeff.=

Apply

Cancel

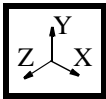
**Define
Element
Properties**

14. Define the Element Properties

Now we will define the element properties for our Quad4 elements. The elements have a thickness of 0.05 inches and should reference the aluminum material property that we have just created.

We will associate the element properties with the geometry model, and not the finite elements. This will allow MSC.Patran to reassociate the properties to the elements if we later decide to remesh the model.

First let's reset the view to make it easier to cursor select our elements. Press Iso 1 View icon from the toolbar:

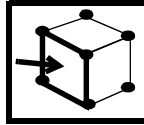


Now let's create our element property and reference it to our geometry model:

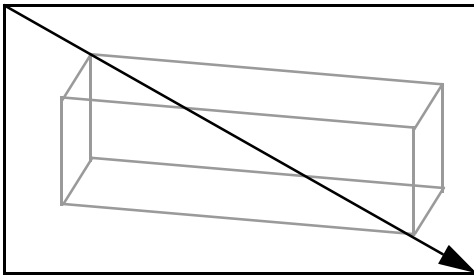
◆ Properties

Action:	<input type="text" value="Create"/>
Dimension:	<input type="text" value="2D"/>
Type:	<input type="text" value="Shell"/>
Property Set Name	<input type="text" value="box_beam_shell"/>
Option(s):	<input type="text" value="Homogenous"/>
	<input type="text" value="Standard Formulation"/> *
	* May not be an option depending on your analysis preference.
<input type="button" value="Input Properties..."/>	
Material Name	<input type="text" value="m:aluminum"/>
Thickness	<input type="text" value="0.05"/>
<input type="button" value="OK"/>	
Select Members	<input type="text"/>

Since we want to associate our element properties with the outside surfaces (faces) of our solid, make sure the Surface or Face icon is highlighted in the select menu:



Now cursor select the entire model (remember our solid is still in the group, "default_group", which is still posted (displayed) to the viewport) Click on the Iso 1 View icon at the top to see the whole model:



Select the entire model using the rectangular selection box.

You should see "Solid 1.1 1.2 1.3 1.4 1.5 1.6" appear in the databox which are the solid face ID's.

15. Create a Load Case

Now we want to create a load case which groups selected loads and boundary conditions into a single set which can be referenced for the analysis.

Create a Load Case

◆ Load Cases

Action:

Load Case Name

☐ Make Current

Load Case Type:

Description

load_case_1

Now click on the Assign/Prioritize Loads/BCs button.

*Select Loads/BCs to add to
spreadsheet*

Disp_fixed

Force_100-pound_down

OK

Apply

16. Perform the Analysis / Check Status of Job / Read Results

Now we are ready to submit our finite element model for analysis.

Under Analysis, we will set up the job for the analysis and submit it directly from MSC.Patran to MSC.Nastran.

◆ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Full Run

Job Name

box_beam_loadcase_1

Translation Parameters...

Solution Type...

Direct Text Input...

Subcase Create...

Subcase Select...

OK

Apply

If you have time, you may want to look at the subordinate forms for these options. The defaults for these options are fine for our model.

load_case_1

After pressing Apply, the heartbeat on the main form will turn blue and you should see a number of messages scroll by in the Command Line. The heartbeat will then turn to green which means you can continue executing operations in MSC.Patran while the analysis is running.

To see if MSC.Nastran has completed with no errors, open another window and search the contents of the file, *box_beam_loadcase_1.f04* in the directory where you started up MSC.Patran for the string EXIT:

For UNIX, enter:

```
% more box_beam_loadcase1.f04 | grep EXIT
```

For Windows NT, enter:

```
% type box_beam_loadcase1.f04 | find "EXIT"
```

If MSC.Nastran completed with no errors, you should see the following line:

```
16:23:52 0:20 20.2 0.0 11.4 0.0 SESTATIC 145 EXIT BEGN
```

Once the analysis has successfully completed, it will produce a *box_beam_loadcase_1.op2* results file. This file must be read into the MSC.Patran database before you can begin to postprocess the results.

◆ Analysis

Action:

Read Output2

Object:

Result Entities

Method:

Translate

Available Jobs

box_beam_loadcase_1

Select Results File...

Available Files

box_beam_loadcase_1.res

OK

Apply

You should see messages appear in the Command Line, stating which results are being read into the database.

Now we are ready to postprocess the results.

17. Unpost (Erase) the Geometry Group.

Unpost
Geometry
Group

Since we are done using the geometry, let's erase or "unpost" the group, "default_group", which contains the geometry part of our model:

Group/Post...

Select Groups to Post

fem_model

Apply

Cancel

Notice the Loads/BCs symbols go away because they are associated with the geometry that is part of "default_group", which is now unposted.

Create a Deformation Plot

18. Create a Deformed Shape Plot

Let's create a deformed shape plot based on the displacement results. This is an excellent way to view the response of our structure. Note: The numbering of the result cases may vary.

◆ Results

Action:

Create

Object:

Deformation

Select Result Case(s)

Select Result

**1.1-DISPLACEMENTS,
TRANSLATIONAL**

Apply

Although there are other ways to reset the graphics display of our model back to the default wireframe, the easiest way is to press **Reset Graphics** icon at the top *before* creating another postprocessing results plot.

Create a Fringe Plot

19. Create a Fringe Plot of the von Mises Stresses.

Next, let's create a color fringe plot of our von Mises stresses. Invariant stresses, such as von Mises stresses, are a good choice for a fringe plot since they are scalar quantities, and they are a good predictor of the yield stress in metals like aluminum.

◆ Results*Action:***Create***Object:***Fringe***Select Result Case(s)**Select Fringe Result***Stress...***Quantity:***von Mises****Apply****20. Create a Combined Fringe/Deformed Plot**

Let's create a contour plot of our membrane stresses on top of a deformed shape plot by doing the following:

◆ Results*Action:***Create***Object:***Deformation****Apply**

Now you should see the contour plot on top of the deformed shape plot.

21. Modify the Finite Element Mode (Optional)

Note the high stress regions are bending stresses in the opposing corners at the box beam's open end (opposite of the welded end). But they are well below aluminum's 30 ksi yield stress.

Being the good engineer that you are, you realize the design could be greatly improved by welding a cap on the open end of the beam.

Try posting the geometry model ("default_group") and mesh the solid face on the open end. You can then quickly equivalence and optimize the model and assign the existing element properties to this new area.

Run the model through MSC.Nastran again and see if the stress levels improve. Good luck!

Close the Database and Quit MSC.Patran

22. Close the Database and Exit MSC.Patran.

File/Quit

Changes to your model will be automatically saved when you close your database or exit MSC.Patran.

Congratulations and give yourself a big pat on the back!

You have just had a glimpse of the future -- the way productive people will be doing finite element analyses from now on.

And there's much more! Try accessing a CAD model, creating varying loads and element or material properties through the use of Fields, interpolating results from one analysis onto another model as loads, and using the Finite Element Sweep mesh creation.

Examples of these and other features of MSC.Patran can be found in Part 10: Example Problems in the MSC.Patran User's Manual.