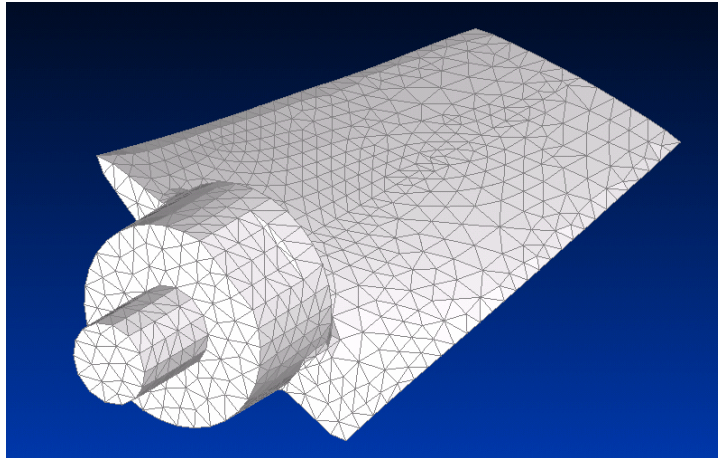


FEMAP Tutorial 1

This tutorial will give a brief introduction to the use of typical finite element software for predicting the deformation, strains and stress in a structure subjected to static loads. The analysis process consists of several steps:

1. Problem Definition (“Pre-processing” Step)
 - a. Definition of the geometry of the structure and the idealization of this geometry for an approximate “finite element mesh.” This geometry definition is very similar to the use of AutoCAD or SolidWorks to describe a geometrical shape.
 - b. Definition of material constitutive properties
 - c. Definition of approximate kinematic boundary conditions
 - d. Definition of the applied loads



**Figure 1: A finite element mesh for a turbine blade.
It consists of three-dimensional finite elements.**

2. Generation of the equilibrium equations relating nodal displacements and forces, $[K]\{q\}=\{Q\}$, and the solution of these equations to obtain nodal displacements and resulting element strains and stresses (the “analysis” or “solver” step).
3. Solution Display (“Post-processing” step)
 - a. Deformed geometry
 - b. Contours plots of various stress components

Typically the pre-processing and post-processing is performed by software that is separate from the solver software. The output of the pre-processor program becomes input to the solver. In turn, the output of the solver program becomes input to the post-processing program. In many cases the solver software is able to perform many kinds of analysis including static and dynamic response prediction, heat transfer analysis, natural frequencies and mode shapes, and other types of structural response. Some analyses are linear while some are nonlinear and take into account nonlinear kinematic and constitutive response.

For this tutorial, we will be using the combinations of FEMAP and NX Nastran. FEMAP will be used for the pre- and post-processing steps while NX Nastran is the solver program. FEMAP stands for Finite Element Modeling and Analysis Program. Nastran was originally developed by NASA and the acronym NASTRAN stands for NASA STress Analysis. Today there are many licensed versions of Nastran that run on computers ranging from PC's to supercomputers. Other analysis programs include ANSYS, ABACUS, ALGOR, PATRAN, MARC, MSC Nastran, etc.

Now, let's consider a bar (or truss member) being pulled by an axial force on both ends. These are often found in trusses, linkages, and similar applications.

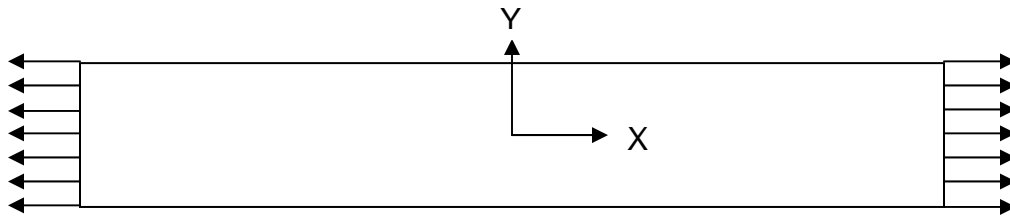


Figure 2: Full bar with axial force

From previous structural knowledge, and from intuition, we can predict that the bar will deform in such a way that it will elongate, and the cross section will uniformly contract due to Poisson's effect.

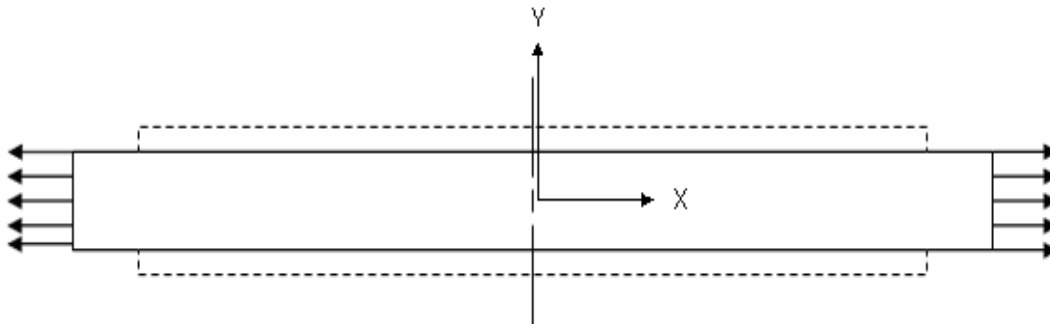


Figure 3: Deformed bar

In order to simplify the model to be drawn in FEMAP, we can utilize symmetry and look at half of the beam, say the right half. The points along the centerline are allowed to move vertically, which allows vertical contraction due to Poisson's effect, and the point at the exact center of the bar is stationary. These kinematic restrictions are called boundary conditions.





Figure 4: Half bar utilizing symmetry

One of the goals of this tutorial is to determine the effects of various ways of idealizing:

1. Boundary Conditions
2. Loading Methods

By analyzing several different cases we will be able to see the impact these various conditions have on the internal stresses and strains in the bar.

Let's define the specific problem we are going to solve:

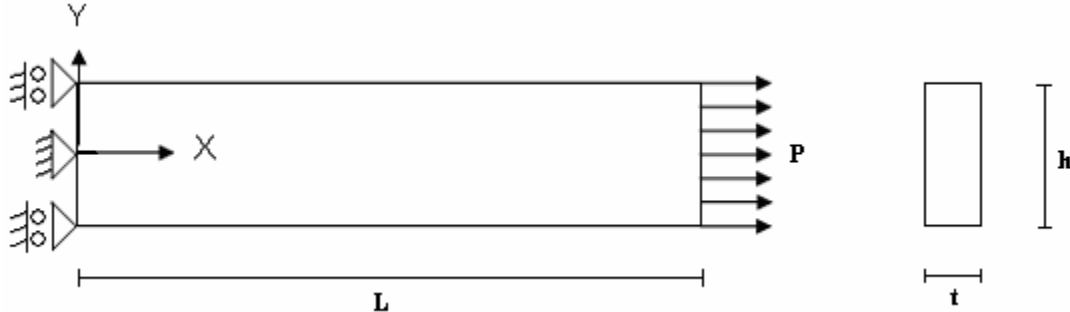


Figure 5: Bar with defined dimensions

The length (L) of the bar is 6 inches, the height (h) is 1 inch, the thickness is 0.25 inches, and the applied traction on the right end is 20,000 psi. This traction represents a total force (P) of 5,000 lbs. The bar is made of AISI 4130 steel which has material properties of Young's Modulus, $E = 29 \times 10^6$ psi, Poisson's ratio, $\nu = 0.32$, and weight density, $\rho = 7.33 \times 10^{-4}$ lb/in³.

Boundary Condition cases:

1. As shown in figure 4, center point fixed and other points allowed to translate vertically on left boundary (simulates Poisson's effect), individual nodes.
2. Each node on the entire left boundary fixed

Loading cases (the traction of 20,000 psi is idealized in two different ways):

1. Distributed point loads along the right boundary
2. Single load on center point of right boundary (another method of idealizing a distributed load)

Using the first boundary conditions and loading case, the steps to develop the model fully in FEMAP will be explicitly described.

For this case, we expect the FE solution will give approximate uniform internal stresses of $\sigma_{xx} = 20000$ psi, $\sigma_{yy} = 0$, psi $\sigma_{xy} = 0$ psi and the total axial deflection will be 0.0043 in. This can be verified from elementary strength of materials (Engr 214).

An important issue before we begin concerns units. Every quantity (number) that you input must be in consistent units. The program makes no assumptions about units. It is up to the user to maintain consistent units. Many FE programs do contain a library of material properties (these are typically in English units; however the user must always be aware of any such units, and provide other input in consistent units).

Let's Get Started:

We will first define the problem to be solved. In order to accomplish this task, we have to describe four types of information about the configuration using FEMAP:

- Geometry – The shape of the model
- Constitutive – What the model is comprised of (materials, properties)
- Boundary conditions – Loads and constraints acting on the model
- Compatibility – How the elements fit together

Explanation of Notation:

- Navigating menus can be troublesome. This tutorial gives the correct menu path using a shorthand notation, which is best described with an example. Consider the phrase

Create.load.nodal

This indicates that one should click the “Create” menu item, followed by the “load” submenu item, and then finally the “nodal” sub-submenu item.

- When a dialog box is to be filled in, the label for each field will be given with the required input in parenthesis. For example,

Young's Modulus, E (30000000)

Indicates that you should put 30000000 in the field labeled “Young's Modulus, E”

- Many times in FEMAP, it is easiest to select a feature of the model on the screen with the mouse rather than typing in its ID number. This will be indicated by underlining the action you should perform. For example,

Geometry.Curve – Line.Points...

Create Line from Points. Select the points X(-3) Y(1) and X(-3) Y(0)

Means that when the ‘Create Line from Points’ box comes up, you should select the points X(-3) Y(1) and X(-3) Y(0) with your mouse (where the number in parenthesis are coordinates, i.e., X(-3) means the point located at $x = -3$).

- Sometimes you must select a checkbox. This will be indicated by **label(check)**, where “label” is the label for the checkbox.
- **Ctrl-Z** is undo if you get into trouble and need to back up. There is no “redo” command
- **Ctrl-D** is redraw, which will clear away deleted features and redraw the model.
- The screen may be zoomed in or out by using the scroll button on your mouse. If you do not have a scroll button, you can use the zoom tool under the view menu, or the magnifying glass buttons on the toolbar (at the top of the screen).
- It is wise to **save the model under different filenames** as you complete major sections. This will allow you to go back to a previous version should you need to. When you are finished, you should have seven or eight files of the model at the different stages of its construction to avoid having to start from a clean slate in order to change one aspect of it. Use a file naming strategy that makes it easy for you to remember what step you were at when you save the file (or, keep good notes!)

FEMAP is located in the Start Menu of the lab PC’s (just like Maple, Microsoft Word, or other software): Start.Programs.Programs.FEMAP v9.0

FEMAP will open and you should see a new workspace, if not got to File.New to open a new workspace.

Define the workplane

Defining the workplane is like creating a sheet of paper that you will draw on. This will specify the location, size, and orientation of the space in which we will create the model. It will give us a frame of reference for our model by creating a grid and visible coordinate axes.

Tools.Workplane (or Press F2 or **Ctrl-W**) to bring up the Workplane Management menu
Snap Options...

Grid And Ruler Spacing.Uniform (check)
Grid And Ruler Spacing.X Grid (0.5)
Grid Style.Dots (check)
Workplane Size.X from (-2) To (12)
Workplane Size.Y from (-3) To (3)
Adjust to Model Size(Uncheck)
Snap To.Snap Grid (check)
OK

Note: what you have just defined is a grid that is 14 x 6. The units of the above entries are in inches.

Define Material Group

As we create our model, we will want groups of materials that we will use to build our structure. To create materials in FEMAP, you simply assign specific material

characteristics (i.e. Young's Modulus, Poisson's Ratio, etc.) to a material ID number. For example, Material ID # 1 might have the material characteristics of carbon steel, and Material ID # 2 might have the material characteristics of 2024 aluminum. Then as we create our model we can specify elements that will be carbon steel and others that will be aluminum. You may define materials by manually entering in the values of those characteristics or by loading a saved material from the material library, as we will do in this exercise. If you load in your own values you must make sure you are using consistent units, i.e. if you enter the material properties in English units, you must also enter the dimensions of the geometry in English units.

Model.Material...

Load...

AISI 4130 Steel (Select from list)

OK

OK

Cancel

Define Property Set

Next, you need to define element property groups that will determine what type of elements will be used to build our model. You will complete this task in the same way that you defined the groups of materials. Property data defines an element's geometry properties (thickness, areas, radii, etc.), specifies mass, and inertia, and it selects a material for the element from the Material Group(s) that you have previously defined. There are many different property types to choose from: line elements (rods and beams), plane elements (membranes and laminates), and 3-dimensional solid elements. The specific type of finite element used will depend upon the geometry of the structure and how it is to be idealized. For example, the bar that is being considered here is clearly three-dimensional, however, it can be idealized as having a two-dimensional state of stress (in this case, two-dimensional in the x-y plane). Although we will be modeling a bar in this exercise, we will be doing so using membrane elements to view the internal stresses in the structure. Membrane elements assume that a plane stress condition acts in the plane of the membrane (again, in this case, the x-y plane). Some pre-processing solvers will call this a plane stress element rather than a membrane element.

Model.Property...

Title (1/4" Steel)

Material (drop down menu select AISI 4130 Steel)

Click Elem/Property Type...

Membrane(check)

OK

Thicknesses, Tavg or T1 (0.25)

OK

Cancel

Model Geometry

Geometry provides the framework for most finite element models. Think about it as if you were drawing a picture of an object. This step is very similar to using AutoCAD, SolidWorks, and similar programs. First, you draw the outline of its shape, and then you "mesh" the outline to better describe it. Here we will use FEMAP's geometry commands

to create the outline, and then later we will “mesh” it with elements. Thus, we will have created our model.

Referring to Fig. 4, recall that we want to define a geometry in the x-y plane that is 6 in. long in the x direction and 1 in. high in the y direction. The following will create a 6” x 1” rectangle.

Geometry.Point...

X(0) Y(0) Z(0) OK

X(6) Y(0) Z(0) OK

X(0) Y(1) Z(0) OK

X(6) Y(1) Z(0) OK

Cancel

Here we create four points that will become the bar, as shown in the schematic below.

View.Autoscale.All (or Press **Ctrl – A**)

Autoscale is a useful command to memorize. It automatically resizes the view of the screen to contain a close view that contains the whole model.

Looking closely you can see cross hairs on the points which you have defined.

There are many ways to create rectangles in FEMAP. Regardless of the technique used, you are really just defining some lines. These lines will be used later to define the regions to be meshed. Meshing refers to the process of dividing up a region into a number of smaller “finite elements.”

Geometry.Curve – Line.Points...

Create Line From Points...

Select a starting point with your mouse by clicking on one of the points with the cross hairs, then select the corresponding point that the line will be drawn to

OK (Hint: you can also double click on the second point to create the line)

Repeat these commands until four lines have been drawn using the four points created. (The lines that will appear between the two points are thin and bluish in color.)

OK

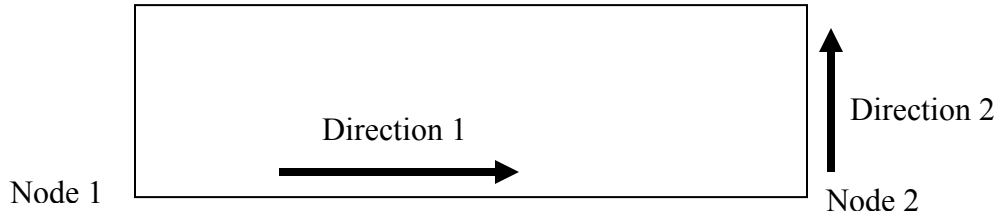
Cancel

Note: There are many ways to create the same geometry. Spend some time in the geometry menu experimenting with different drawing methods such as drawing a rectangle directly from the line commands, or from the rectangle command, etc.

Creating the Mesh

Now that the shape of the model has been created, we will use the ‘Mesh.Between...’ command, which subdivides a rectangle by making divisions along the first and second directions defined. The directions referred to are determined by the first and second

node specified (direction 1) and the second and third node specified (direction 2). This is demonstrated in the figure below:



Mesh.Between... (or **Control – B**)

Node And Element Options.Property(1..1/4" Steel) *This selects what type of elements should be used*

Mesh Size.#Nodes.Dir1(21) *(This means that the length in direction 1 will be defined by 21 “nodal points,” or 20 elements).*

Mesh Size.#Nodes.Dir2(5) *(This means that the length in direction 2 will be defined by 5 “nodal points,” or 4 elements).*

OK

X(0) Y(0) Z(0) OK

Direction 1 is from (0,0,0) to (6,0,0)

X(6) Y(0) Z(0) OK

Direction 2 is from (6,0,0) to (6,1,0)

X(6) Y(1) Z(0) OK

X(0) Y(1) Z(0) OK

Cancel

Note: If the snap option is not set to Snap to Point, right click anywhere on the screen, and select Snap to Point on the pop-up menu that appears. Now, you may easily and accurately select the points with your mouse instead of entering them manually.

We have now divided the 6” x 1” plane area into a 20 x 5 finite element mesh, which is shown in Fig. 5

If you have not already done so, this would be an ideal place to save the model.

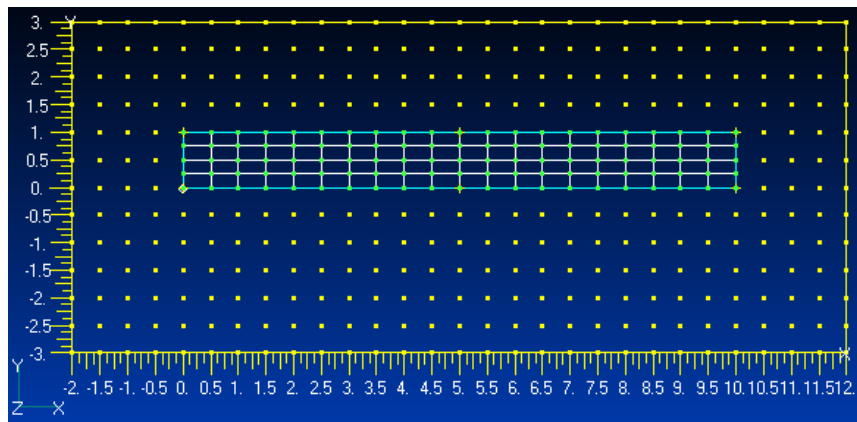
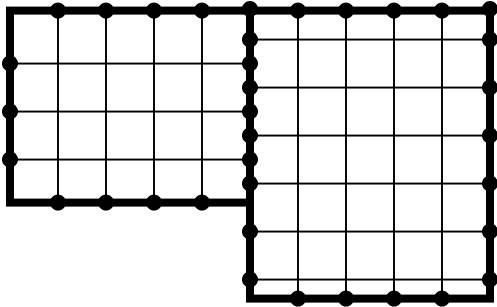


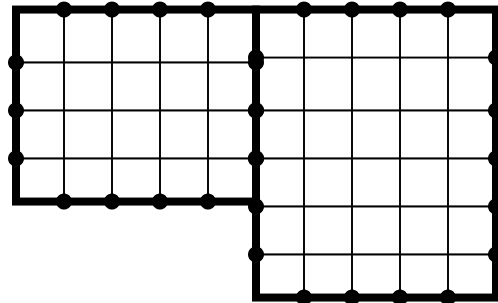
Figure 6: Meshed Bar

Compatibility

Note: This section is not applicable for our current geometry, but it is an extremely important issue within FEMAP, thus it will be covered here for future reference. It is applicable if our geometry includes more than one section, such as:



These two superelements would **NOT** be compatible. The nodes of the abutting superelement sides do *not* coincide



These two superelements would be compatible. The nodes of the abutting superelement sides *do* coincide, but it is **dangerous to match partial edges.**

Often a complex geometry such as that shown above includes multiple super elements, which must be combined into one structure to satisfy our compatibility requirement. This is because each super element has its own set of nodes. Therefore, the common boundary (line) between the two super elements will have two sets of nodes (one from each super element) that each occupies the same point in space. However these common nodes must be converted to one unique node point so that compatibility is achieved. To accomplish this task, we use the “Check Coincident Nodes” command, which combines nodes that occupy the same location (“coinciding” in the same physical place). By combining these nodes, the superelements are then “connected” by these shared nodes. If you neglect to combine the nodes, it will be easy to notice that in the analysis results, the model will be disjointed along those boundaries, i.e., it is as if the two super elements are not connected.

To perform this function, perform the following steps:

- Tools.Check.Coincident Nodes...
- Entity Selection.Select All
- OK
- OK to specify additional range of nodes to merge? Yes
- Entity Selection.Select All
- OK
- Options.Merge Coincident Entities (check)
- OK

After you complete this step, a message should appear on the gray bar below the main window informing you how many nodes were merged

Loads and Constraints

In general both the loads and constraints are applied the same way in FEMAP. They are the same types of loads and constraints that can be applied when we do finite element analysis by hand calculations. To apply a load or a constraint to a model, you first must create a “set” in which the load or constraint will be defined. This allows you to change the loading or the constraints on your model very easily and quickly by simply defining a new set and only applying that set in the analysis. Once the load and constraint sets have been defined, you may apply any of them in a variety of ways. In this exercise, we will constrain and load individual nodes, as well as geometry curves (i.e. any line previously defined in the geometry).

Create a constraint set

Model.Constraint.Set...(or **Shift-F2**)

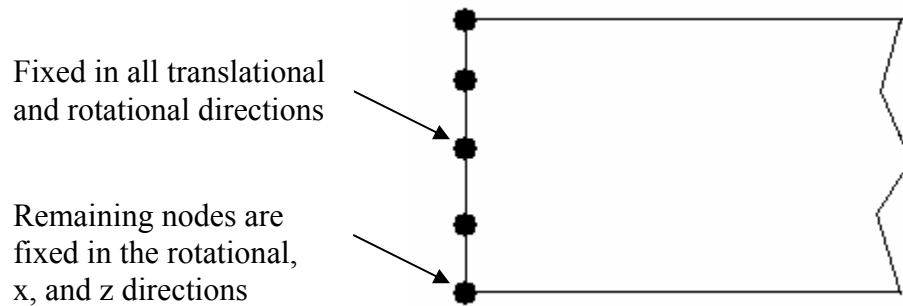
Title(ConstraintSet1)

OK

Note: Choose a title that reminds you of what kinematic constraints are being applied to the structure, or keep good notes on what each constraint set consists of.

Specify the constraints

For this model we will constrain the center node on the left boundary in all directions of translation and rotation. The rest of the nodes we will constrain in the x and z directions and all directions of rotation. This will simulate the effects of Poisson’s ratio by allowing the model to effectively ‘shrink’ along its vertical cross section.



Model.Constraint.Nodal...

Entity Selection.Select the center node by clicking on it with you cursor

OK

DOF.Fixed

fixing translation and rotation in all directions

OK

Entity Selection.Select the remaining nodes

OK

DOF.TX, TZ, RX, RY, RZ (check) *fixing translation in the x and z direction*

Cancel

and rotation in all directions (which means that the node can move in the y direction)

Create a load set

Model.Load.Set...(or **Control-F2**)

Title(LoadSet1)

OK

Specify the loading

*For the first case we will approximate the traction of $T = 20,000$ psi by a set of point forces at the right boundary. The total axial force is $P = T * A = 20,000$ psi ($1'' \times 0.25''$, = 5,000 lbs.) Since we have five nodes on the right boundary, we will place 1,000 lbs on each node of the right boundary. In loading case 2 we will model the distributed load as a point load on the center node of the right boundary (case 2). This will allow us to see how internal stresses are affected by a point load versus a distributed load.*

Model.Load.Nodal...

Entity Selection.Select the five nodes on the bar's right side

OK

Load.FX.Value(1000)

This places a 1000 unit load in the x-direction at the node(s) selected.

OK

Cancel

This will model a 5000 unit distributed load.

At this point, the finite element model has been completed. If you have not already done so, make sure you save the model before analysis (it will be saved as a .mod file). It is wise to save it under a different file name. The next step is to invoke the solver or analysis program, which will create and then solve the equations of equilibrium $[K]\{q\}=\{Q\}$.

Model Analysis

NX Nastran analysis using FEMAP generated models

While the model file could be exported to any solver, we will be using NX Nastran. NX Nastran is a special version of NASTRAN marketed by UGS (see <http://www.ugs.com/products/nx/simulation/advanced/nastran/> for further details). The first step is to select NX Nastran as the analysis program, and to also specify that a static analysis is to be performed.

Model.Analysis

Analysis Set Manager...

New

Analysis Set...

Title (Case 1.1) *Note: The case title can be named whatever you want it to be.*

Analysis Program (36..NX NASTRAN) *This indicates that we will use NX Nastran as the solver (analysis program)*

Analysis Type (1..Static)

OK

DONE

To start the analysis step, click on File.Analyze

File.Analyze

The NASTRAN analysis window will now automatically open and run. You will see the status of the analysis process in the “Analysis Monitor” window on the left of the screen. When Nastran has finished, you will see a message displayed here stating “NX Nastran finished” followed by the date.

Post-Processing

Post-processing takes the raw data returned by the solver and converts it into visual and quantitative results such as displacements, stress contours, and stress criteria.

Stress Contours

View.Select... (or Press **F5**)

Deformed Style.Deform(check)

Contour Style.Contour(check)

Deformed and Contour Data...

Output Set (Select most recent output set – it will be the last one on the menu)

Output Vectors.Deformation(1..Total Translation)

Output Vectors.Contour(7433..Plate Bot VonMises)

OK

OK

Stress Contours will give you a view of the stress distribution throughout the structure, but they do not supply specific details

The final VonMises contoured model should look similar to this:

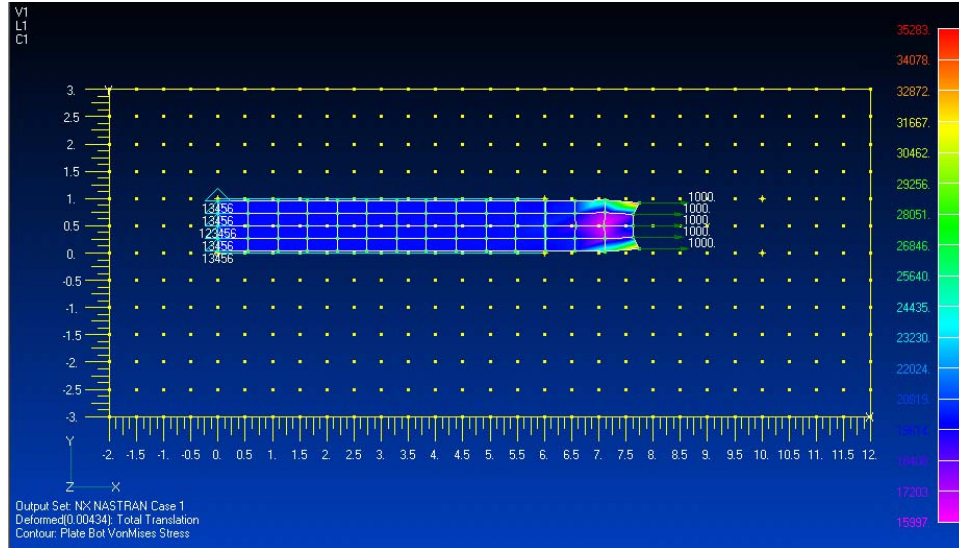


Figure 7: 10% Scaled VonMises Stress

Average Bottom VonMises stress is a special stress component that takes a two- or three-dimensional stress state and reduces it to a single value. Another single value that could be plotted is a “principle stress.” There is no difference in average top or bottom VonMises stress in this application (since the stress does not vary in the z direction). It is important to understand the difference in deformation and contour. The selection for deformation will physically be the values that change the shape of the model. It makes sense to deform the model by translation, and although it can be done, it makes less sense to deform the model by stress values. Contour is the color that is plotted on the model. The stress values are usually what we contour on the model. In this particular problem, it may have been better to display T1 translation (which is translation in the x direction), rather than the Total translation (which is the sum of the x, y and z translation components). You may want to try this and see the difference.

Several items to note:

In the “Select PostProcessing Data” window, when you make the different selections for deformation and contour, note that the values to the right of these drop down menus show the max and min values for the choice selected, and the element where that value occurs.

In the menu View.Options, there are many different options that you can set to view the output. The most important option to note is that FEMAP has a default scale factor for its output of 10% to 1. This means the model appears more deformed than it actually is. To set the scale to a one to one ratio:

View.Options

Category (Post Processing)

Options (Deformed Style)

Scale % (1)

Scale Act. (1)

OK

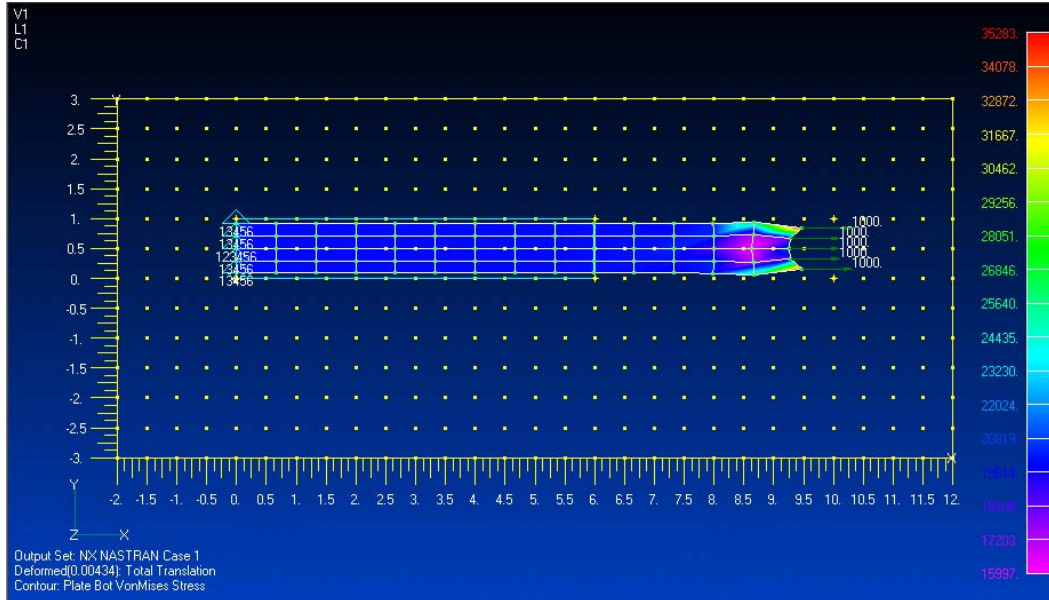
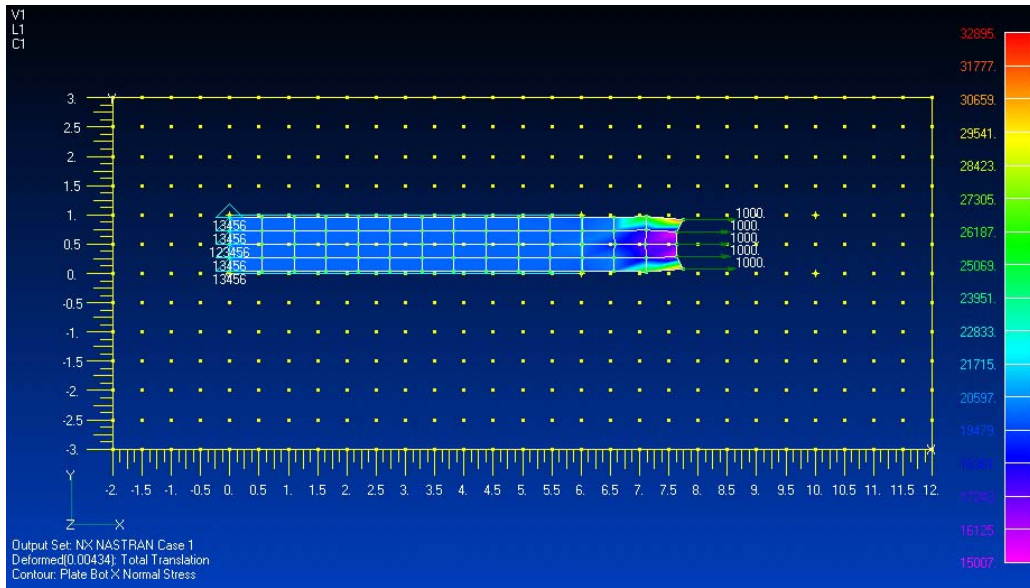


Figure 8: 20% Scaled VonMises Stress

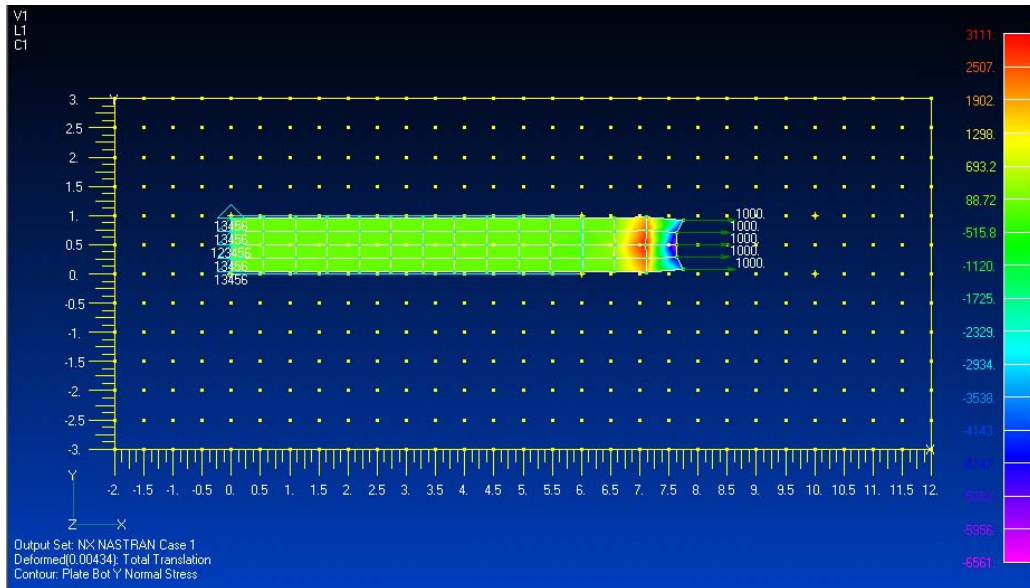
Notice in this 20% scaled view you can clearly see Poisson's effect. In most problems, the deformations are usually very small (in fact, when you look at a physical structure under load, you will most likely not be able to observe the deformation since it is so small). Hence, it is customary to scale the deformations and make them large enough so that they can be seen on a deformed drawing. So kept this in mind when you look at the deformed geometry – it is typically exaggerated so that you can view deformation more easily.

We can repeat the above procedure to view several other stresses including axial stress in the x and y direction, as well as the shear in the x-y plane. When contouring the shear stresses, make sure that you choose the correct plane to view it in. If the model is set up so that you are viewing the x-y plane, but you choose x-z shear stress the contour will make very little sense.

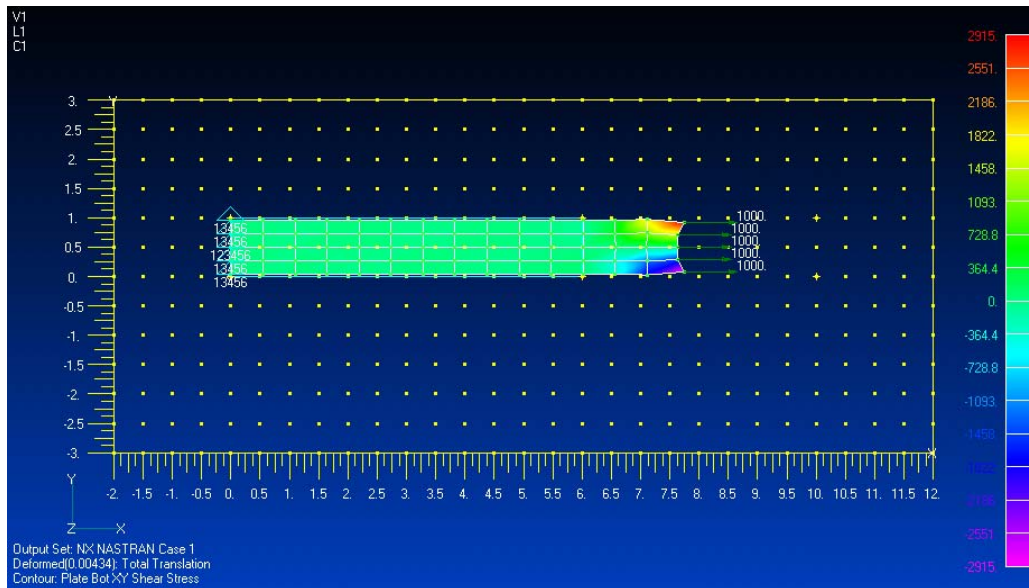
The final contoured axial stress in the x direction should look similar to this (Plate Bot X Normal Stress):



The final contoured axial stress in the y direction should look similar to this (Plate Bot Y Normal Stress):



The final contoured shear stress in the x-y plane should look similar to this (Plate Bot XY Shear Stress):



Stress Criteria

View.Select... (or Press **F5**)
 Deformed Style.Deform(check)
 Contour Style.Criteria(check)
 OK

Stress Criteria will label each element with its average or maximum stress value. These criteria are useful in determining specific stresses in an element.

View Options... (or Press **F6**)
 Category.Postprocessing(check)
 Options (Select Contour/Criteria Style)
 Data Conversion (1..Maximum Value)

This option is normally preset to show average stress, but you can also view maximum stress in each element.

However, if the stress state of a specific element is needed, queries may be performed to retrieve this data.

List.Output.Query...
 Output Query...
 Output Set (Select most recent output set)
 Category (4..Stress)
 Entity.Elem(Check)
 Entity.ID(Enter element ID or place cursor in field and select element)
 More *This prints the query but allows you to immediately request another query*
 OK *If you are finished with queries*

FEMAP will now print the stress state of that element in the active output set in the gray bar at the bottom of the screen.

Another useful tool for determining nodal or elemental values of various quantities can be achieved by using the “Selector Entity” button in the top toolbar. This button is on the top row of toolbars just to the right of the middle of the top row (move your mouse over these toolbar icons and a description will be displayed). From the icon drop down box, you can then select things like node, element, material, load, etc display. Suppose you select “element” by click on it. Once you select element, you can move your mouse over the model, click on an element of the model, and the various information for that element will be displayed in the “Entity Editor” window on the left of the screen (element node numbers, property ID, element type, aspect ration, stress components, etc.). Give this a try, and select a different item with the “Selector Entity” toolbar icon.

Other Solutions (with different load and constraint cases)

Now, we will repeat this exercise using the different load cases and constraint cases outlined previously.

Boundary Condition cases:

- 1. As shown in figure 4, center point fixed and other points allowed to translate vertically on left boundary (simulates Poisson’s effect), individual nodes.*
- 2. Each node on the entire left boundary fixed*

Loading cases:

- 1. Distributed point loads along right boundary*
- 2. Single load on center point of right boundary (another method of idealizing a distributed load)*

To create a different load set:

Model.Load.Set...(or Press **Ctrl-F2**)

ID (2)
Title(LoadSet2)
OK

Model.Constraint.Set...(or Press **Shift-F2**)

ID (2)
Title(ConstraintSet2)
OK

An alternative way to create new constraint and load sets is to click on the gray bar at the bottom of the screen where it says “Ld: 1” and “Con: 1.” Click on Set... and enter a new set.

Follow the steps previously outlined to apply constraints and loads on the model with the new conditions.

For the remaining cases use the first BC and analyze each loading case, then use the second BC, and analyze each loading case again.

The following steps are used to perform another analysis on the same model with different boundary and loading conditions.

Model Analysis

Analysis Set Manager...

New (name a new Analysis Set "Case1.2")

Click the plus sign next to Analysis Set 2

Click the plus sign next to Master Requests and Conditions

Click the Plus sign next to Boundary Conditions

Here you can double click on the Constraints or Loads and choose which set you would like analyzed

Highlight the new analysis set you would like analyzed

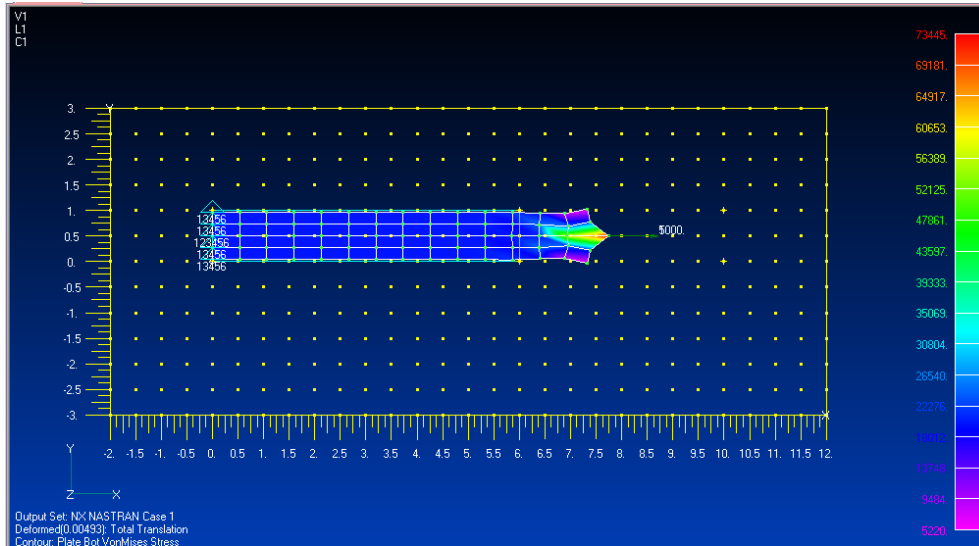
Click Analyze

When you go to View>Select to see the new output, under the deformed and contour data button, make sure you choose the most recent "Output Set."

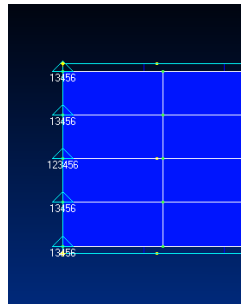
The following are results from the various analyses. Only the VonMises and Shear stresses are shown. Make sure you look at the axial X and Y stresses also. What major differences do you see?

Constraint 1, Load Set 2

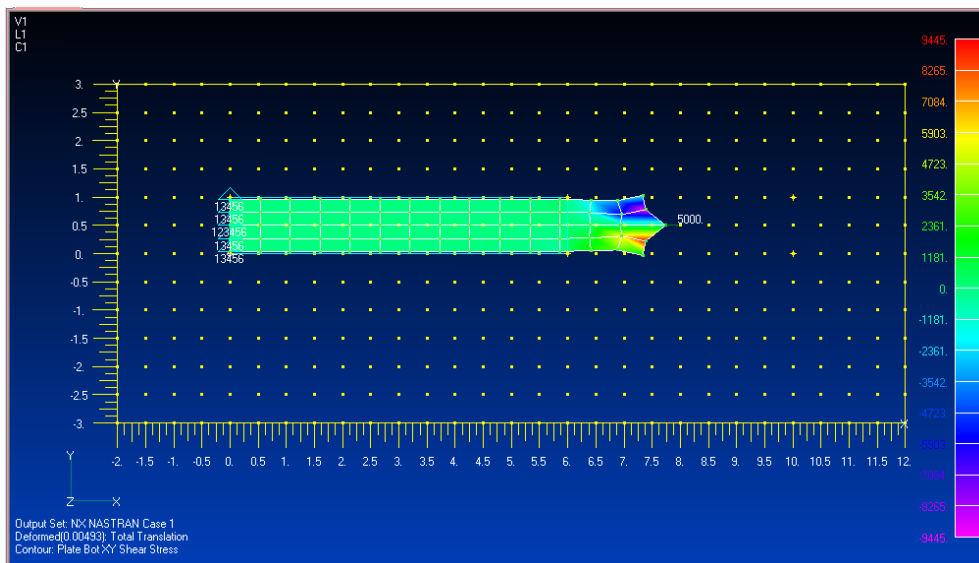
The VonMises contoured model should look similar to this:



Note the appearance of the left boundary.

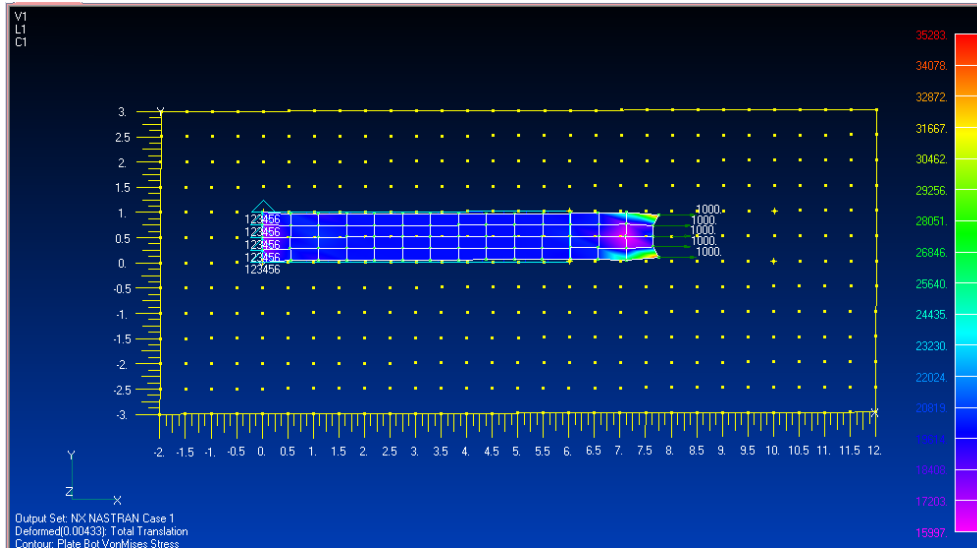


The XY Shear Stress contoured model should look similar to this:

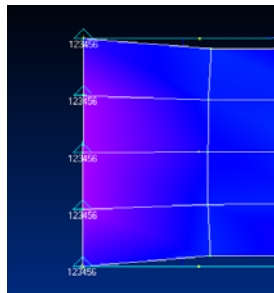


Constraint 2, Load Set 1

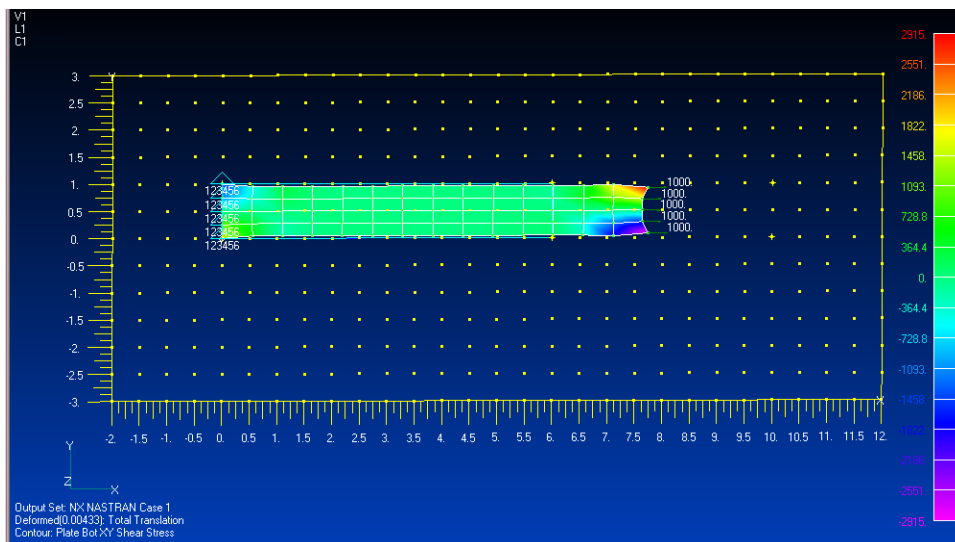
The VonMises contoured model should look similar to this:



Note the difference in the left boundary. Does this change the values of the stresses?

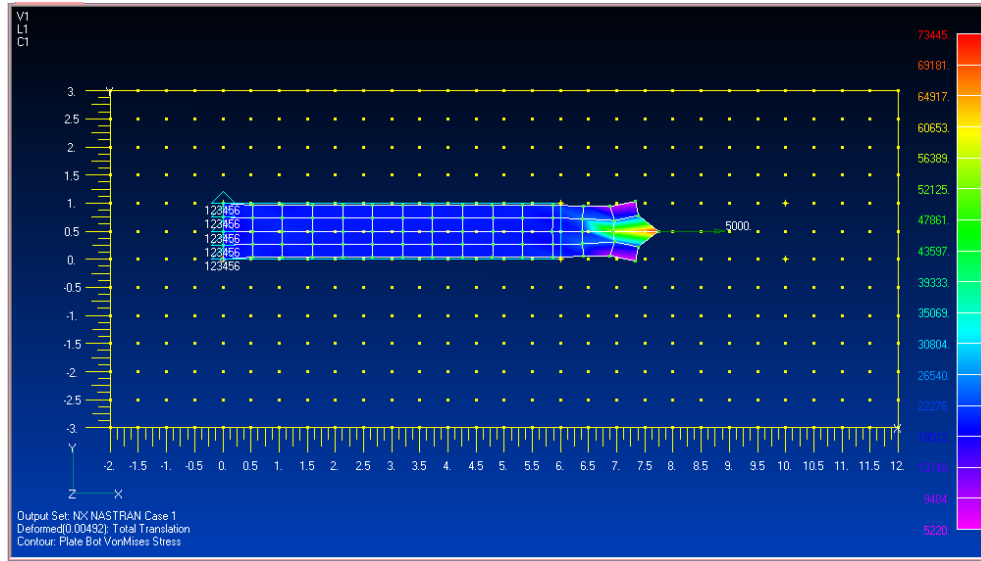


The XY Shear stress contoured model should look similar to this:

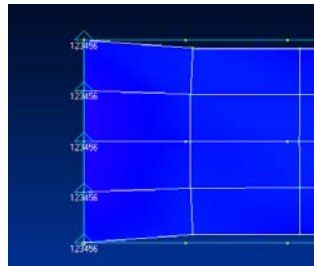


Constraint 2, Load Set 2

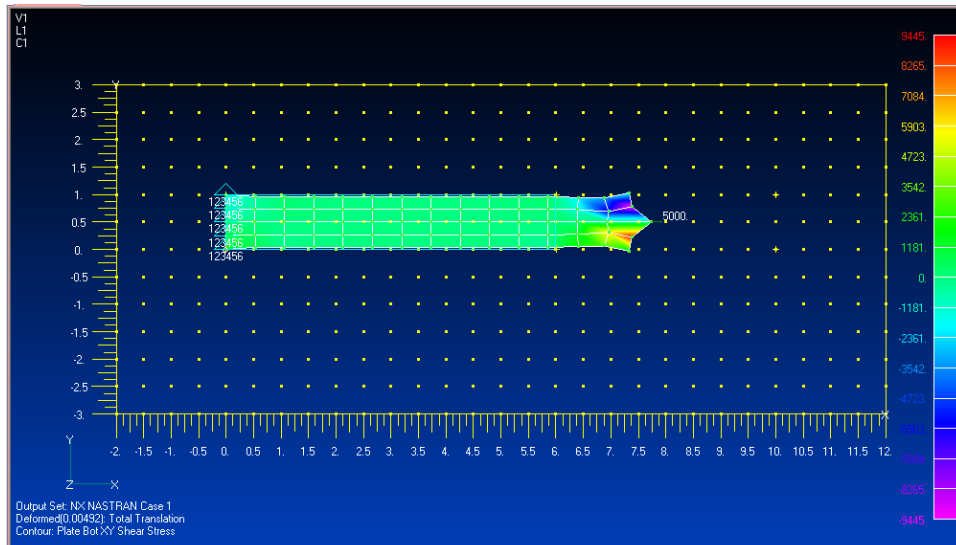
The VonMises contoured model should look similar to this:



Notice the difference in the left boundary, which is “fixed” and not allowed to contract. Does this change the values of the stresses?



The XY Shear stress contoured model should look similar to this:



If you would like to look at animated solutions to several three-dimensional problems, you might take a look at the following link:
<http://www.ugs.com/products/nx/simulation/advanced/nastran/index.shtml#animations>