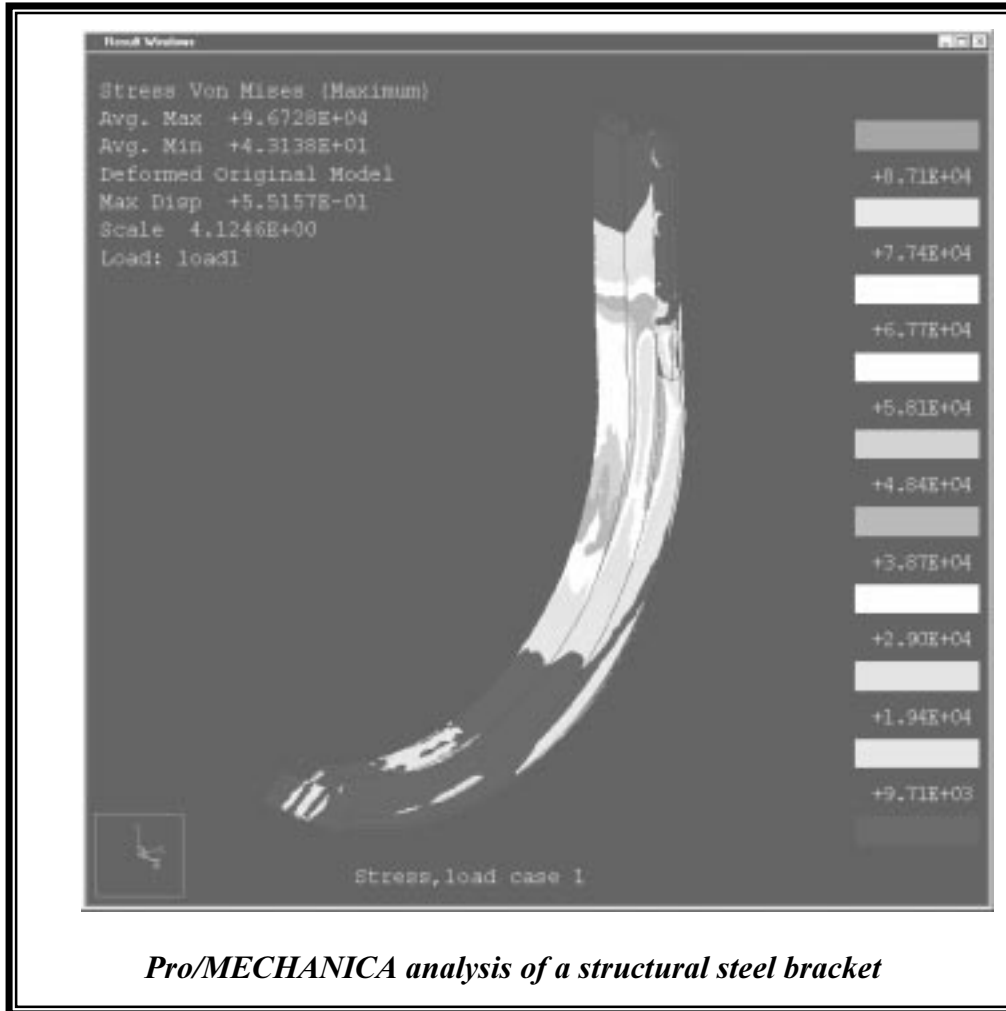


PRO/MECHANICA 2001 STRUCTURE: ELEMENTS AND APPLICATIONS

Integrated Mode



Yves Gagnon, M.A.Sc.
Okanagan University College

SDC
PUBLICATIONS

Schroff Development Corporation

www.schroff.com

Exercise 1: Beam Elements

Objectives

At the end of this exercise, the learner should be able to:

1. Set up a model using standard cross-sections beam elements with proper orientation;
2. Set up and run a static analysis;
3. Understand the units used in Pro/Mechanica®;
4. View maximum VM stress and maximum displacement results;
5. Create shear and moment diagrams;
6. Understand the basic solving principle of the Finite Element method.

Introduction

Beam elements are fast and efficient elements in FEA. They are 1-D in nature but still represent what is called a 3-D idealization. A 3-D idealization is a FEA modeling perspective of the model. For a beam element, imagine an I-beam. It could be represented as a straight line, with a cross-section the shape of an “I” assigned to it. This is, in essence, how a beam element is represented.

From a modeling perspective, beam elements are relatively easy to create. They are defined by:

- A datum curve determining the position of the end points of the beam;
- A material;
- A cross-sectional area (which will give an area moment of inertia and a torsional stiffness)
- A defined orientation for the cross-section.

Each beam element has its own internal coordinate system (let’s name it the Beam Section Coordinate System or BSCS). The BSCS has the following characteristics in Pro/Mechanica®:

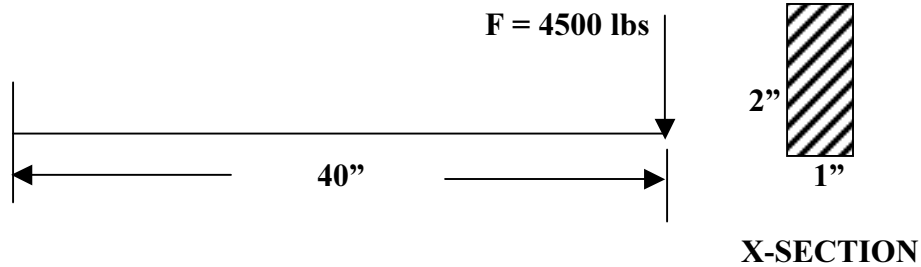
- The x-axis is along the length of the beam (the beam’s axis);
- The y-axis defines the *perpendicular* orientation of the beam section with respect to the beam’s axis;
- The z-axis will follow according the first two axis’ above.

The restriction of using beam elements in Pro/Mechanica®, as with any other software is that the section remains perpendicular and planar to the datum curve (the beam axis) through the entire FEA solution. Before using beam elements, make sure that the part looks similar to a beam and that the aspect ratio (l/t) > 10 .

Procedure

(Note: Before you start, create a directory named: beam using explorer)

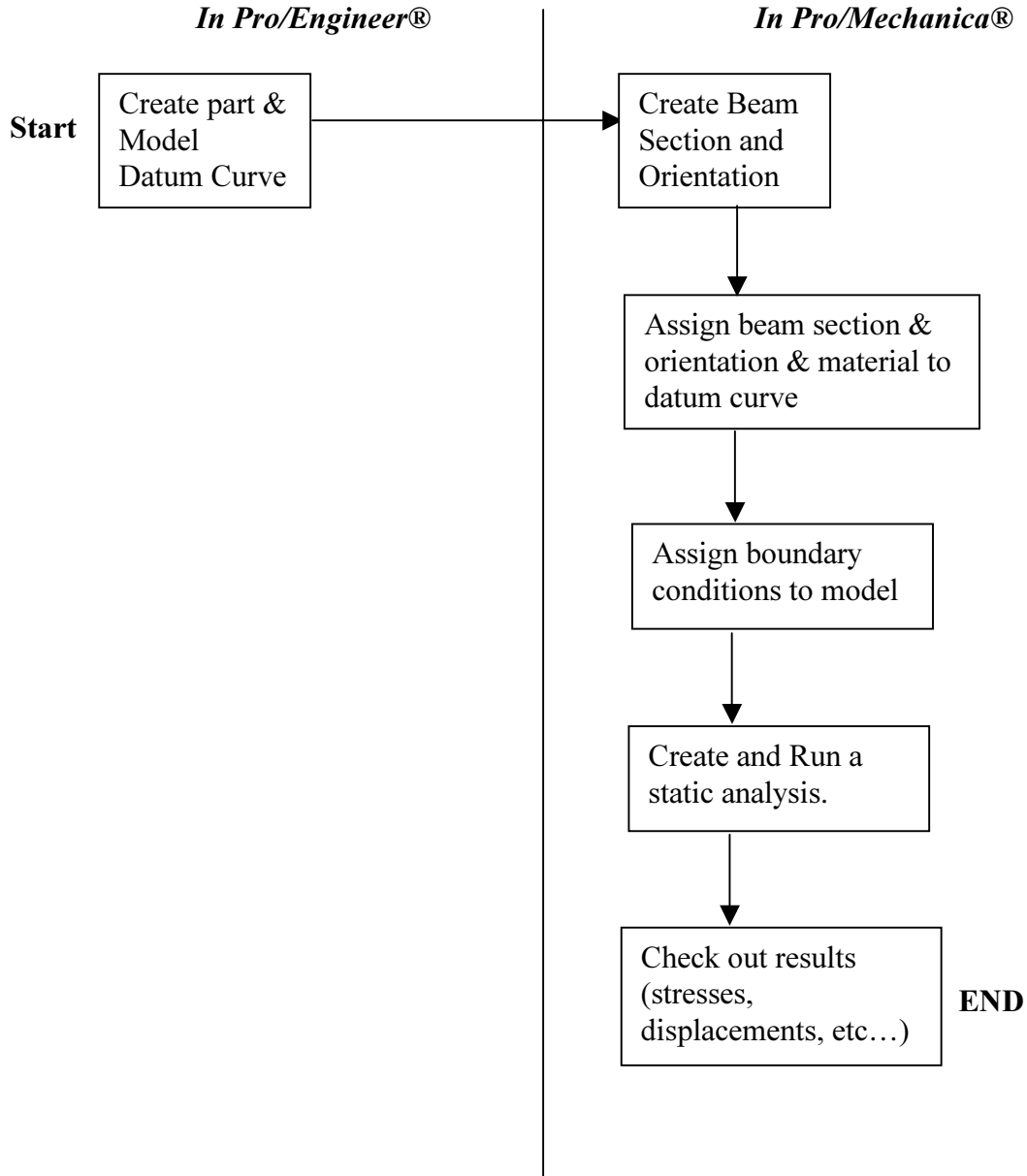
The problem to be solved:



Objectives: Draw Shear and Moment diagrams, and find the Maximum VM stress and maximum deflection.

Flow Chart of Procedure

Here is an overview of the exercise, in the form of a flow chart, showing the different steps involve in the analysis.



➔ 1. Start up Pro/Engineer 2001

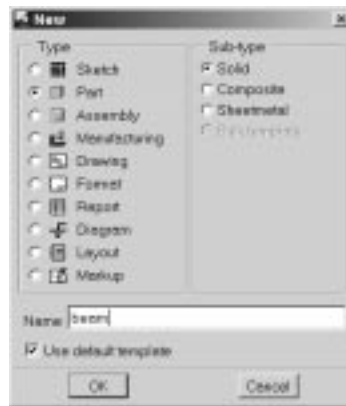
Details on how to this is different from system to system. For a typical windows platform:

Start > Programs > Pro/Engineer2001 > Pro/Engineer
(It takes 30 seconds approximately for Pro/Engineer to start.)

Set up your working directory. (**File > Set Working Directory**)
Select your working directory, then click on **Accept**.

➔ 2. Create a part named: beam

Select **File>new**



Select part and type in: **beam**; then Click on **OK**.

➔ 3. Setting up in Pro/ENGINEER for FEA modeling

Before we go to Pro/Mechanica, we must understand how beam elements are created. Think of beam modeling as a sweep-type protrusion in Pro/Engineer. Except in the case of FEA modeling, we will create a datum curve in Pro/Engineer, then assign the cross-section (the beam element) in Pro/Mechanica.


➔ 3.1 Creating a datum curve for beam assignment

Click on the  icon.

From the right menu, select: **sketch > Done**.

Use *front as your sketching plane* and accept the default viewing direction; click on **OK**.

Select top from the right menu and select the *top plane as the top reference* for the plane.

Once in sketcher, Use the line sketching tool  icon and sketch a *horizontal datum curve* as shown below with the following characteristics:

- 40 units long;
- Aligned with the top plane, symmetric about the right plane.

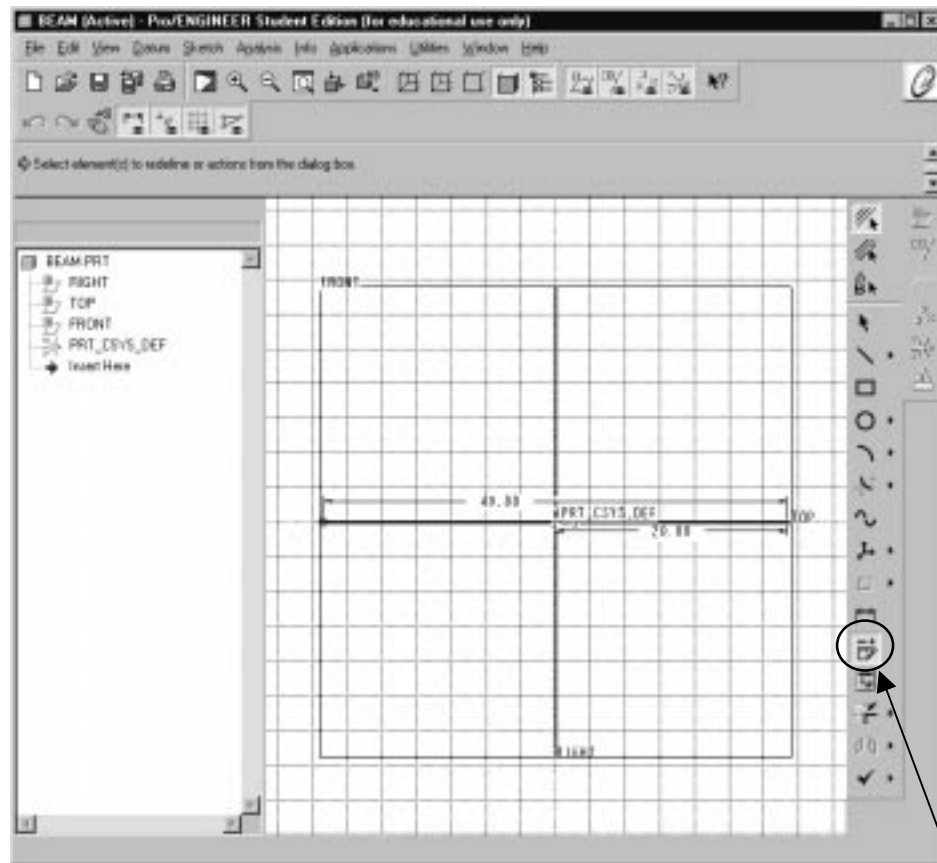



Figure 1: View of sketch for datum curve

Note: To modify your dimension, select the dimension to be modified and click on **this icon**. (or alternatively **double click** on the dimension)

Select **done** out of sketcher ( icon) and **click on OK** off the dialog box. Turn datum planes off using the datum icon at the top of your screen.

(Pro/Mechanica portion)

➔ **1. Go to Pro/Mechanica from Pro/Engineer**

Select **Applications>Mechanica** from the top pull-down menu.

(Wait a few seconds)

You will see a unit info dialogue box come up on the screen as shown below:



Important: Have a look at the working units and then Click *Continue* to accept the units message.

Select *Structure* from the right menu.

➔ **2. Beam Assignment**

2.1 Beam Section

Select **Model>idealizations>Beams>Sections**. The following dialogue box will come up.



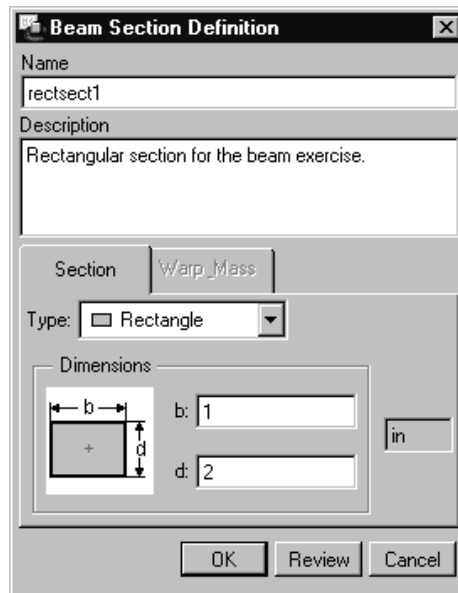
Click on **New**.

The dialogue box shown below will come up. Enter the information as seen below.



STOP

What we are doing at this time is giving the cross-sectional dimensions of the beam element used for the analysis. At this time you must know these dimensions. The cross-section is constant through the datum curve representing the beam. The beam for our analysis is 1 in wide and 2 inches high.



Once the information is entered, click on **OK**. Then **close** the beam section dialogue box. Then select **Done/Return**.

Note: if the section is not in the above menu of pre-defined X-section, you can sketch it easily using the sketch options off the type menu. The menu picks are 'sketch thin' or sketch solid. It will then take you to the Pro/Engineer sketcher.



STOP

We must now orient the vector of the cross-section. Even though it will show up on the screen if we don't, Pro/M wants to make sure that it is oriented the proper way. The software needs insight here as to which way to go and to make sure that we know what we are doing!

A vector will be used to determine the orientation. This vector will determine the beam section coordinate system orientation.

2.2 Beam Orientation

Select **Model>idealizations>Beams>Orientations**

Once the Beam orientation dialogue box comes up, click on **New**. Enter the following information in the box.



Click on **Ok** then **close** the beam orientation dialogue box.

2.3 Beam Definition

Select **New** off the beams menu. The following beam definition window comes up. Enter and select information as shown.

Under Name: Enter **beam1**.

Under references: Select **Edge/Curve**

Under edges, **click on the arrow** and **select the datum curve** on the screen, then click on **done sel**.


(pay attention, selecting it more than once reverses the selection here.)

Material: Select **Steel**.

Under type: select **More...**, the following window will come up:



Select **steel** off the left column and **move it to the right** column by clicking on the triple arrows box in the center. Then click on OK.

Defined by: Y direction of **vector in WCS** Vector (0, 0, 1) (this defines the orientation of the cross-section, as explained on the previous page in the *stop* note.) 

Select section and orientation as shown. There are no beam releases for this analysis.



Select **OK – Done/Return** and note that ***cyan rectangles*** appear along the datum curve. Zoom in on one of the rectangles and look! You can see a vector inside the beam. This vector is oriented in the coordinates given above (0, 0, 1). You could easily change the orientation of that vector simply by changing the coordinates of the vector.

Our beam idealization is fully defined. Good job!


➔ **3. Constraints**

Select **Model > Constraints > New > Point** from the right side menu.

The following dialogue box will come up. Enter and select the following information:

Name: End_constr_1

Member of set: ConstraintSet1

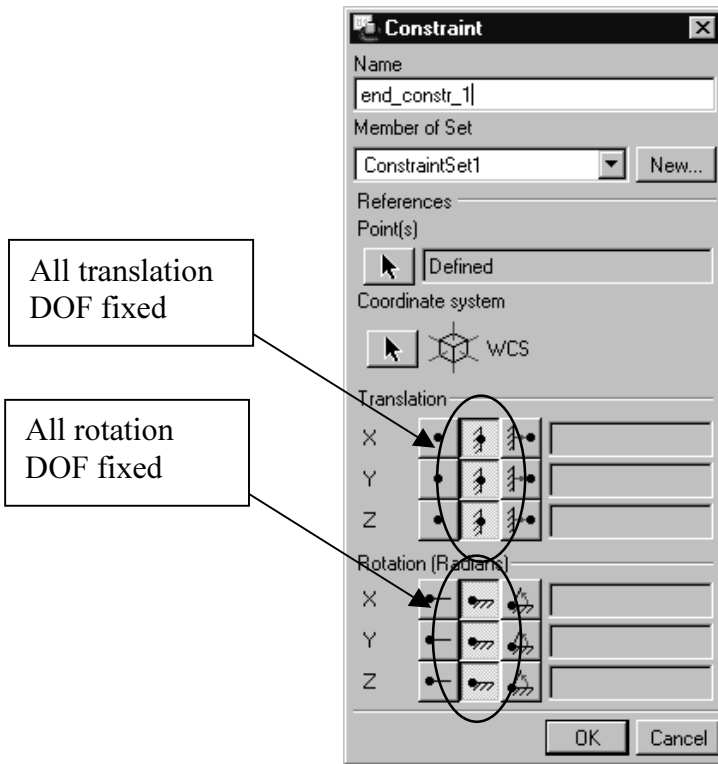
Under points, click on the arrow icon. 

From the datum points menu that comes up, Select **Create > On vertex** and click on the **positive X-axis** end of the beam.

Click on **Done select** and then **Done** off the right menu. (or middle mouse click twice)

A point should then show up at the positive x-axis end of the beam.

Nothing further is needed for this box. We accept the defaults for the WCS and keep all degrees of freedom fixed.



Select **OK** off the constraint dialog box. Select **Done/Return** off the right side menu.

→ 4. Loads

Select **Loads > New > Point**

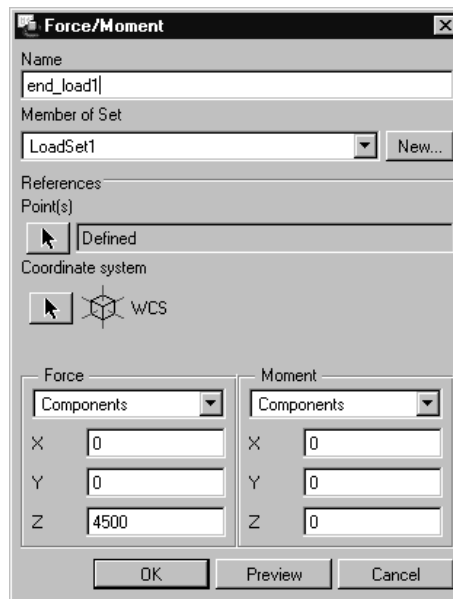
The following dialogue box will appear. Fill out the information as seen below.

Under reference-points, click on the arrow icon. 

From the datum points menu that comes up, Select **Create>On vertex** and select the **negative X-axis** end of the beam.

Click on **Done select** and then **Done** off the right menu. (or middle mouse button click twice)

Under Force-components, enter 4500 in the Z direction box.

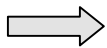
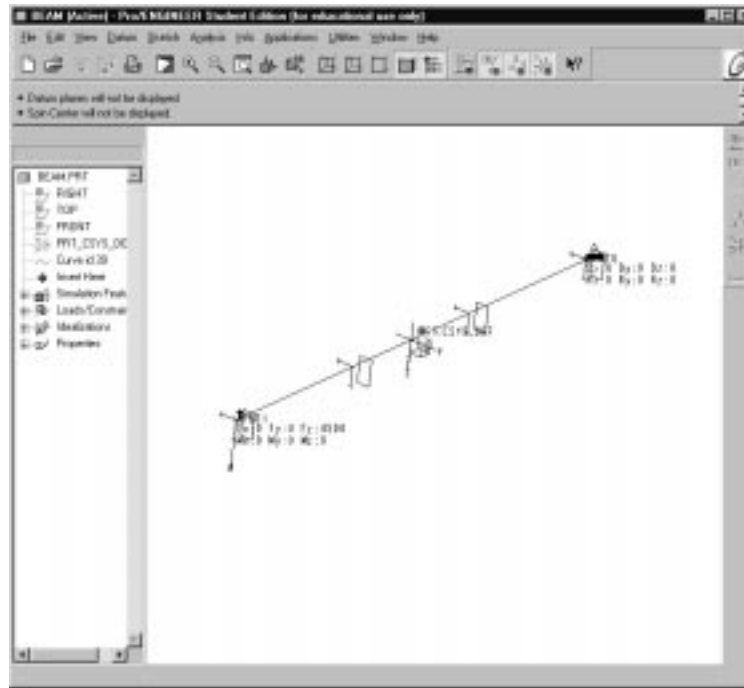


Select **OK** off the constraint dialog box. Select **Done/Return** off the right side menu.

In order to properly show the load, from the *top pull-down menu*, select **View>Simulation display>Settings** and click on **tails touching** under load arrows. Also, click on **value** under Load/Constraint display.

Click on **OK** to close the window.

The resulting window should look like the following:



5. Set up and run Analysis

5.1 Static analysis

Select **Analyses** from the right menu and select **Static > New** from the analysis form. The following dialogue box will come up. Enter the following information:

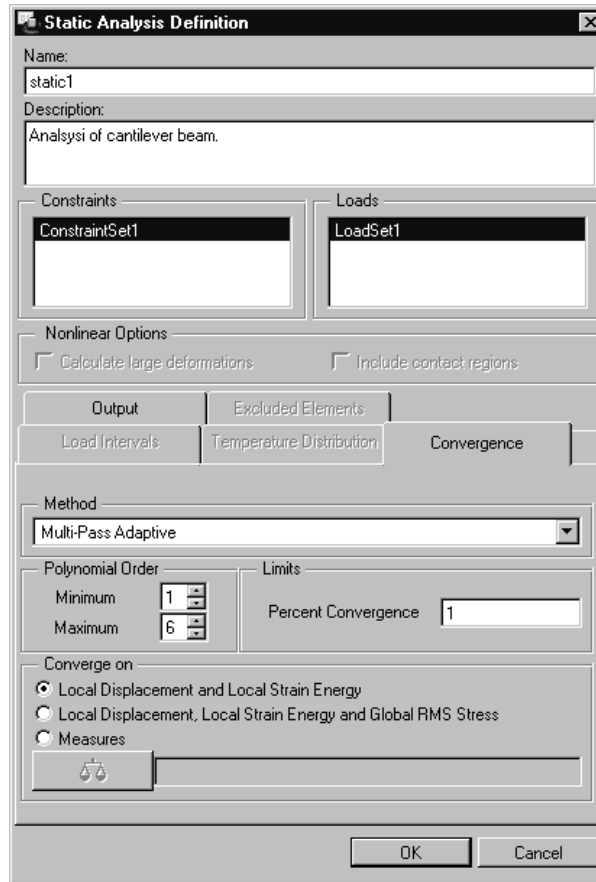
Name: Static1

Description: Analysis of cantilever beam.

Constraints and Loads as shown below.

Method: Multi-Pass Adaptive with 1 percent convergence (since beams are fast and accurate)

Converge on local displacement and strain energy.



Click on **OK** and then **Close**.

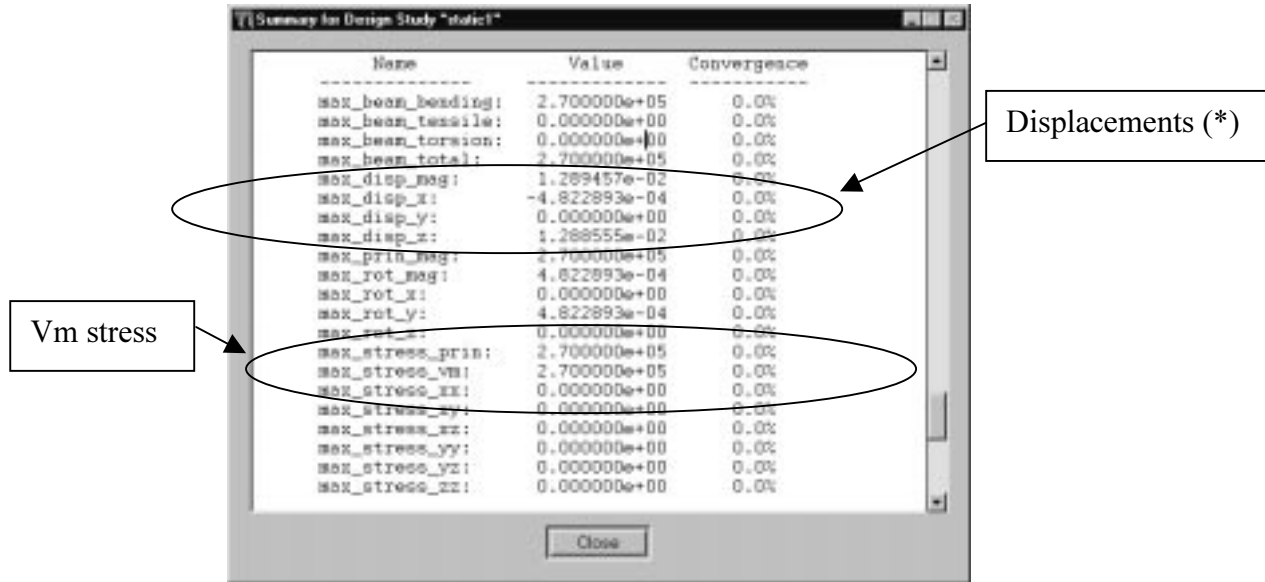
5.2 Run Analysis

Select **Run > Start**. The analysis will now be running.

The software will ask you if you want error detection (yes / no)? Answer **yes**.

To check things as the calculations are performed, click on **Summary**.
(Note how fast the solution converged)

From the summary file, the following information is important to consider.



We are getting a maximum bending stress of 2.7 e+05 psi or 270 ksi. Which is the same value as the exact solution.

What you should look for from the results of this analysis:

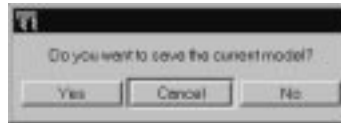
Item	Description	Where to find it	What it should be
Convergence	The convergence will let us know if all elements converged and if they all converged to a solution.	Click on Summary under the Run menu and scroll down	For beams, less than 1% (0% for this analysis)
Maximum VM stress or displacement of Model	Gives the maximum Von Mises Stress or displacement for the analysis	Click on Summary under the Run menu and scroll down	270 ksi (max_stress_vm)
			.012885 (*) (max_displ_z)
VM stress or displacement over entire model	Gives the Von Mises Stress for the analysis for the entire model	See procedure below	Depends on boundary conditions and geometry

Note: (*) Be careful with the *units of displacement* in the results given by Pro/Mechanica®. This is discussed later in the exercise. (See p. 23)

Select **Close** and **Done**.

6. Analysis results

Select **Results**. The following box will come up. Answer yes.



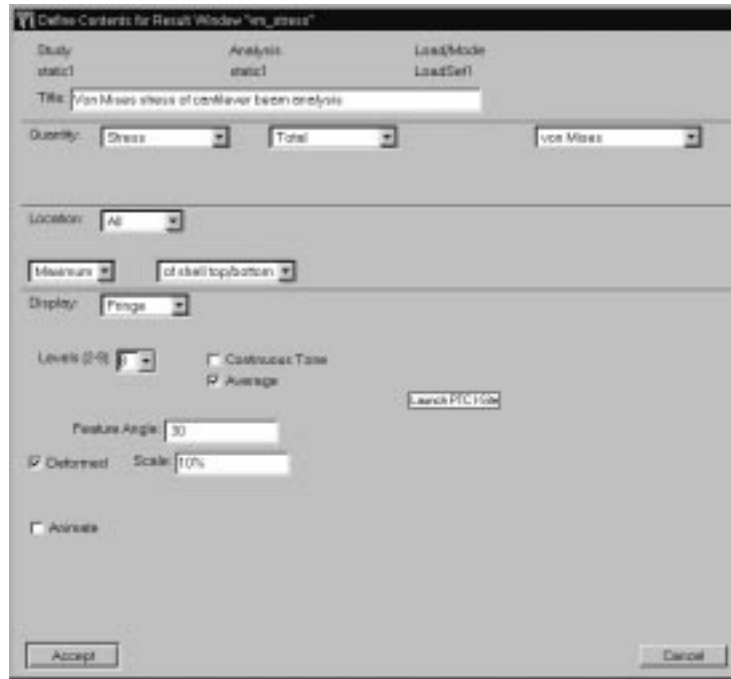
Once the empty window comes up, **Click** on this icon named 'insert a new definition'.



The following box will come up. Type in **vm_stress** (for Von Mises stress)



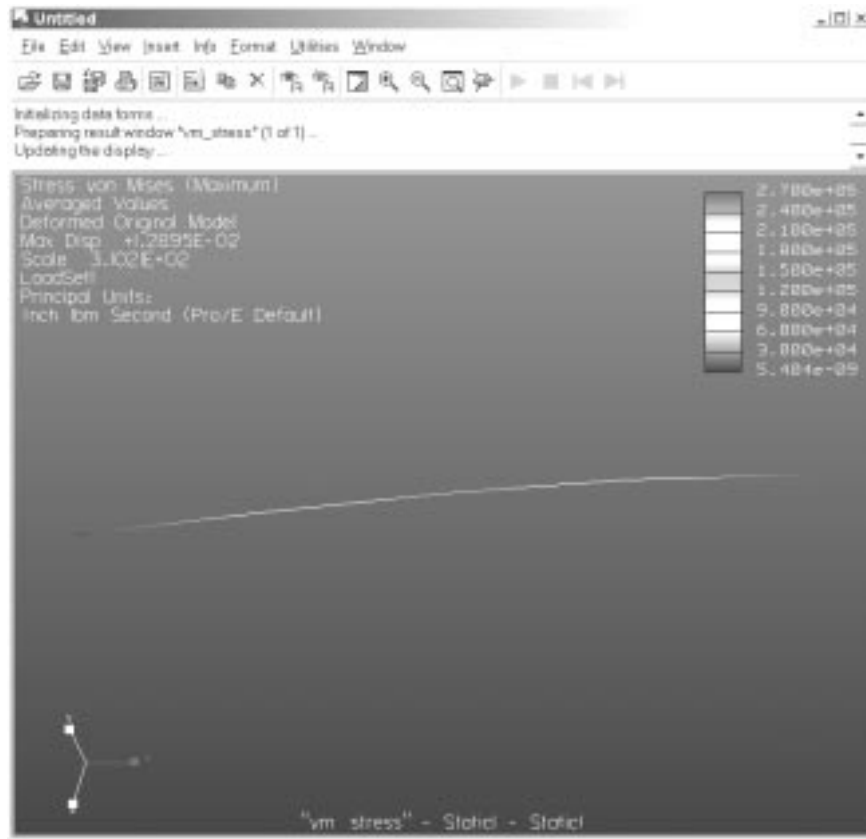
Select **static1** for the analysis. Then click on **Accept**. The following contents definition box will come up. Select and type in all information as shown below.



Click on **Accept**. From pull-down menu, select **view > display**. The following window will come up:



Select **vm_stress** then **OK**. Then the following results window will come up:

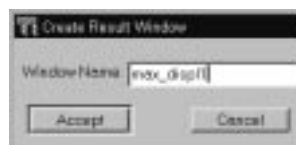


The results from a beam analysis are not the most impressive to look at. But they are accurate, providing your geometry and boundary conditions are precise. The only colors that we see are along the beam axis. We can see that the maximum stresses are at the fixed end of the beam. Note that the maximum value is the same as the one given in the summary file.

Note: you can save this window by simply selecting the save or save as icon on the top of the window.

Select **File/Exit Results** and answer no to the question: Do you want to save the current window?

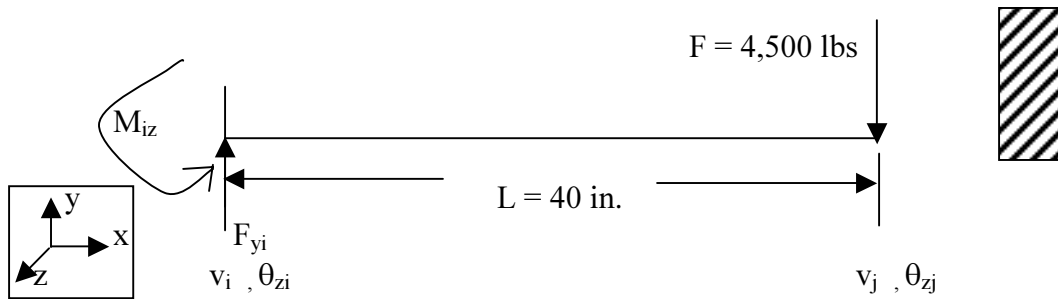
As an exercise, create the following result window in order to look at the maximum displacement from the model. Simply follow procedure 6 above, the only changes being under quantity on the definition form.



Approximate Finite Element Analysis Solution (Theory)

(Don't skip this, as there is some more Pro/Mechanica® procedures included with it to make it more fun)

Solving the beam analysis with FEA theory, the problem can be seen as follows:



Symbol Definition:

Symbol	Definition
L	Length of beam
M_{iz}	Bending moment at the fixed end of the beam
F_{yi}	
E	E is the young's modulus of elasticity of the material.
I	the 2 nd moment of Area
d^2v/dx^2	the curvature of the beam
v_i	The vertical deflection at $x = 0$.
θ_{zi}	The beam slope at $x = 0$.
v_j	The vertical deflection at $x = L$.
θ_{zj}	The beam slope at $x = L$.

The general equation for beam deflection is from beam theory:

$$M_z = EI_z \frac{d^2v}{dx^2}$$

where;

$$M_z = F_{yi}x + M_{zi}$$

Consequently;

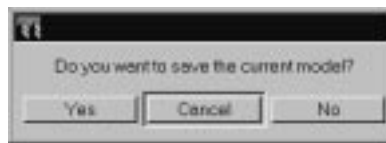
$$EI_z \frac{d^2v}{dx^2} = F_{yi}x + M_{zi} \quad (1-1)$$

Let's get the moment diagrams from Pro/Mechanica®, this will be fun:

From your beam model, where you are in **Applications > Mechanics > Structure**, select **Results** > then **click on the insert a new definition icon**.



The following box will come up.



Answer **yes**.

Select **create** and then the following box will come up. Type in **shr_moments** (we will show both diagrams on the same window.)



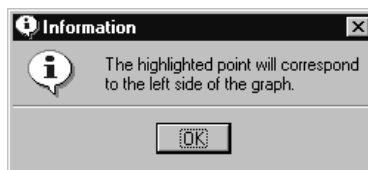
Click on **Accept**.

Select **static1** for the analysis. Then click on **Accept**. The following contents definition box will come up. Select and type in all information as shown below.



Enter and select the following information as indicated:

- **Title:** Shear and Moment Diagrams
- **Quantity:** Select shear and moment (and de-select P, Vz, Mx and My as they are not necessary for this problem). Refer to the coordinate system on the beam model for those. We only need The shear along the z-axis and the moment with respect to the y-axis. (if you are not sure that you followed the modeling properly for this exercise, you can keep all shear and moment selected, it is not going to change the answer.)
- **Location:** Make sure that **beams** is selected. **Click on Select and select the beam off the screen.** You will then be prompted the following message as Pro/Mechanica® highlights one of the ends of the beam.



➔ Make sure that you look at the coordinate system presented to you when you get this message as it affect the de-selection of the 2nd bullet point above,

We have no choice to accept this message, we must note where the point is though as it will help us interpret the diagrams below. Click on **OK**, then **Accept** then **Show**.

The result window should look as follows, where:

- The maximum shear is 4,500 lbs and;
- The maximum bending moment is 180,000 in.lbs.

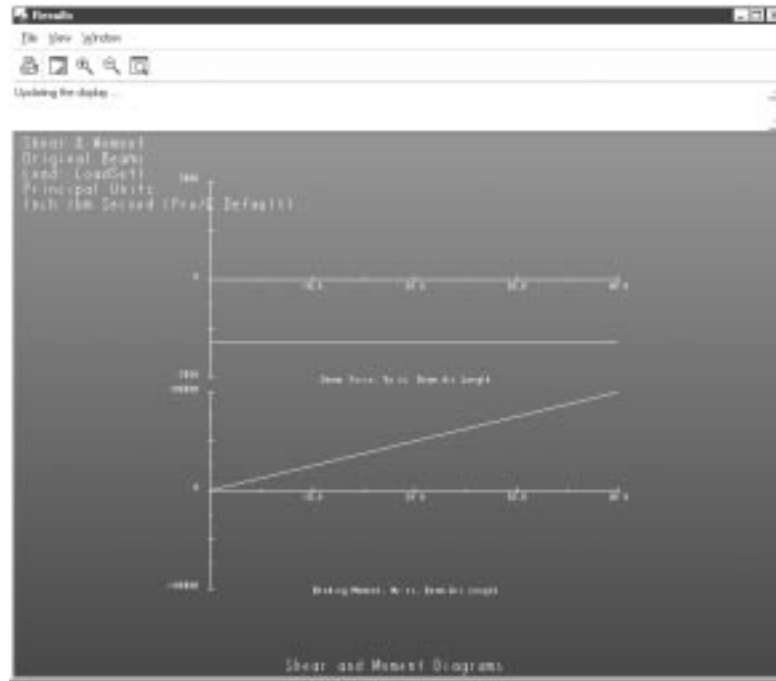


Figure 2: Shear and moment diagrams

Select **File > Exit Results**.

From the diagrams above, it looks like Pro/Mechanica® has taken the point where the load was applied as the reference point. We must understand where the reference point is, as it affects the shape (not the actual values) of the diagrams.

Let's integrate twice (Yahoo!) equation 1-1 above. This will give the following:

$$EI_z \frac{dv}{dx} = \frac{F_{yi} x^2}{2} + M_{zi} x + A_1 \quad \boxed{1^{\text{st}} \text{ integration}}$$

$$EI_z v = \frac{F_{yi} x^3}{6} + \frac{M_{zi} x^2}{2} + A_1 x + A_2 \quad \boxed{2^{\text{nd}} \text{ integration}}$$

From our problem definition, we have the following boundary conditions:

At $x = 0$	At $x = L$
$v_i = 0$	v_j
$\theta_{zi} = 0$	θ_{zj}

Where v is the deflection (vertical) and θ is the slope at the respective locations.

Which will give the following 2 equations for the bending moment and the reaction force at the fixed end of the beam (i):

$$M_i = \frac{6EI_z}{l^2} v_j + \frac{2EI_z}{l} \theta_{zj} \quad (1-2)$$

$$F_{yi} = \frac{12EI_z}{l^3} v_j + \frac{6EI_z}{l^2} \theta_{zj}$$

Which can be written under the following matrix form: $\{K\} \{D\} = \{F\}$ as follows:

$$EI \begin{vmatrix} \frac{12}{l^3} & \frac{6}{l^2} \\ \frac{6}{l^2} & \frac{2}{l} \end{vmatrix} \begin{Bmatrix} v_j \\ \theta_{zj} \end{Bmatrix} = \begin{Bmatrix} F_{yi} \\ M_{zi} \end{Bmatrix}$$

Where $\{K\}$ is the stiffness matrix for our beam deflection problem:

From our diagrams and figure above, we have the following data:

- $F_{yi} = 4,500$ lbs and;
- $M_{zi} = 180,000$ in. lbs
- $E = 29 \times 10^6$ psi
- $I = bh^3/12 = (1) (2)^3 / 12 = .667$ in.³

What we are looking for here is the value of v_j , solving the matrices above using the Gauss elimination method for v_j gives a maximum displacement of 4.95 in.

Do you remember how to solve matrices?

Let's bring back those bad memories:

We have from equation (1-2):

$$\left(\begin{array}{cc|c} \frac{EI12}{l^3} & \frac{EI6}{l^2} & 4,500 \\ \frac{EI6}{l^2} & \frac{EI2}{l} & 180,000 \end{array} \right) \begin{array}{l} \boxed{\text{Line 1}} \\ \boxed{\text{Line 2}} \end{array}$$

We need to bring the matrix to a form as follows:

$$\left(\begin{array}{cc|c} 1 & A1 & C1 \\ 0 & 1 & C2 \end{array} \right)$$

So let's perform the following operations on the matrix:

- Divide line 1 by $EI12 / l^3$ and;
- Divide line 2 by $EI6 / l^2$.

Which gives the following matrix:

$$\left(\begin{array}{cc|c} 1 & l/2 & \frac{l^3}{EI} 375 \\ 1 & l/3 & \frac{l^2}{EI} 30,000 \end{array} \right)$$

Let's now *subtract line 1 from line 2*.

$$\left(\begin{array}{cc|c} 1 & l/2 & \frac{l^3}{EI} 375 \\ 0 & -l/6 & \frac{l^2}{EI} 30,000 - \frac{l^3}{EI} 375 \end{array} \right)$$

Finally, let's divide line 2 by $-l/6$

$$\left(\begin{array}{cc|c} 1 & l/2 & \frac{l^3}{EI} 375 \\ 0 & 1 & -\frac{l}{EI} 180000 + \frac{l^2}{EI} 4,500 \end{array} \right)$$

Which gives the following solution, when all proper values are put in the matrix:

$\theta_{zj} = -.186 \text{ rd and}$

$v_j = 4.95 \text{ in.}$

Comparing the deflections:

FEA approximate solution	Pro/Mechanica® solution	Exact Solution ($PL^3 / 3EI$)
4.95 in.	.0129 (sec^2)	4.96 in.

The above results do not concur. Well, the software ones anyway. What went wrong here? Pro/Mechanica® actually gives a deflection value much smaller than the exact and FEA approximation solutions. But the maximum Von Mises stress calculated by Pro/Mechanica® was accurate?

The problem is with the system of units that Pro/Mechanica® uses. We need to multiply the answer we got for deflection by the gravitational (386.4 in/sec^2), so the final Pro/Mechanica® solution answer will be 4.98 in.

Conclusion

Beam elements are fast and accurate elements. They are mainly used by analysis for getting an idea of the big picture of an assembly of different cross-section. The frame rail of a trailer for instance. They are also useful to get the shear and moment diagrams of structures.

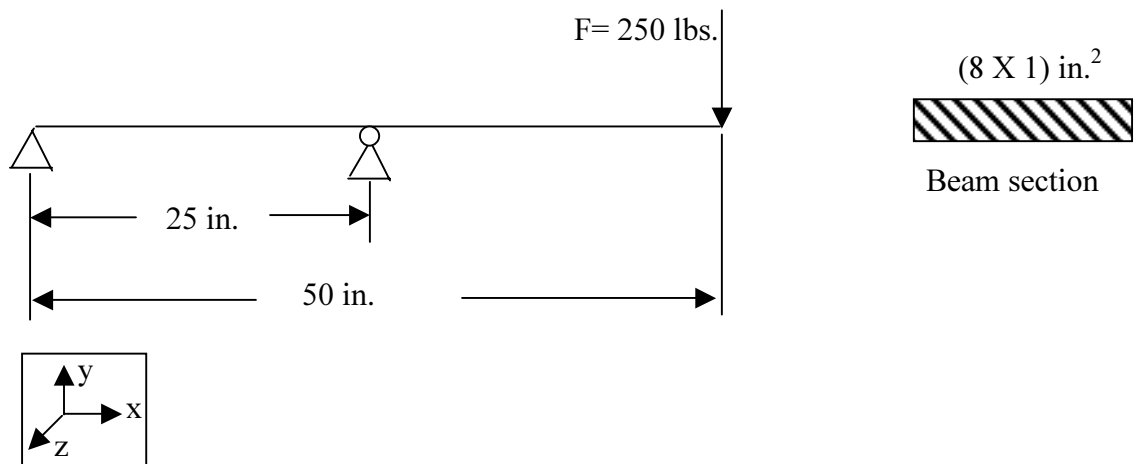
The modeling is fast (datum curves) and the beam assignment is also fast. Remember: If it looks and smells like a beam, then use beam elements for the analysis. On the other hand, you will not be able to study stress concentrations around holes for instance, shell elements should then be used for this type of analysis.

Be careful with the units of displacement in the results given by Pro/Mechanica®, multiply your answer by the gravitational constant in order to get to the units that you are looking for (inches in this case).

Project 1


Problem Definition

Determine the location and magnitudes of the maximum VM stress and the maximum displacement (in y) of the following application (diving board).



Notes and Hints:

 Fixes the location in all 6 DOFs.

 Fixes the y direction DOF and the rotation with respect to x axis.

Material: Use Aluminum 2024

Results

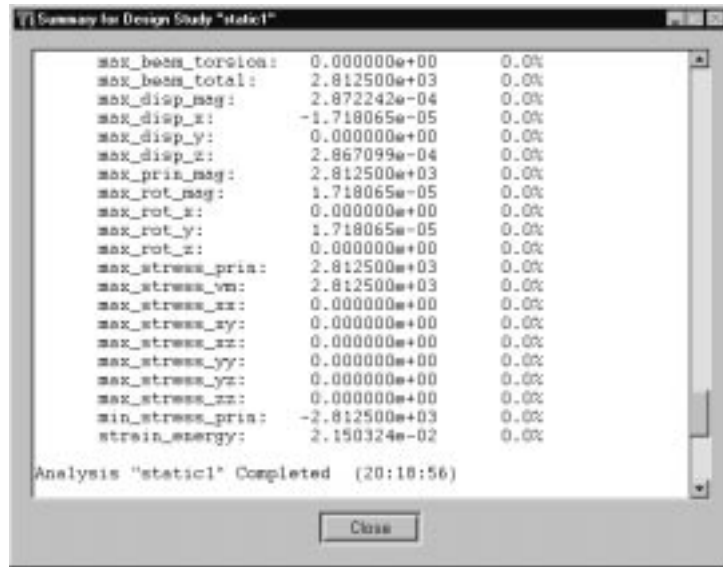


Figure 3: Summary file results for project1

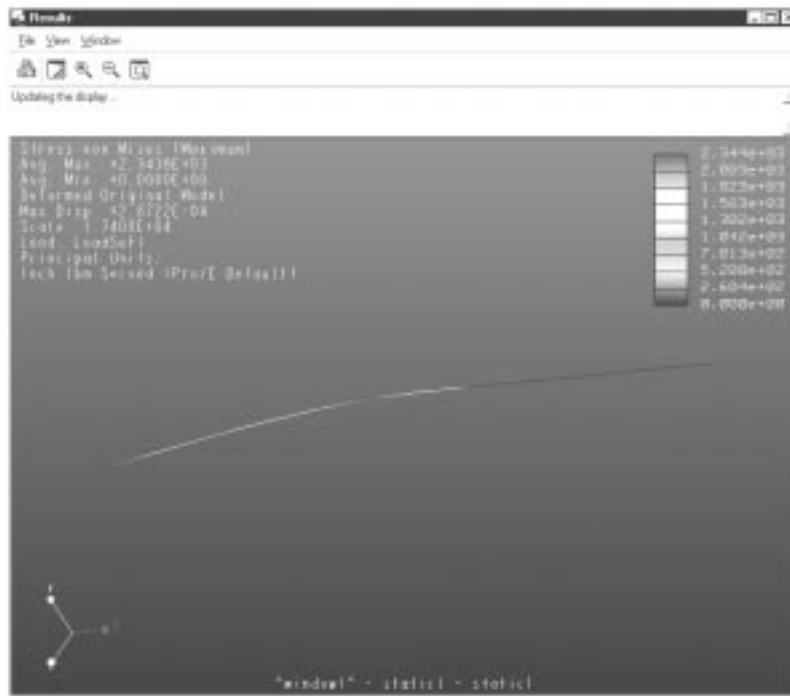


Figure 4: Von Mises Stress fringe plot for project1

References

1. Toogood, Roger, *Pro/Mechanica Structure Tutorial (release 2000i)*, SDC Publ., 1999.
2. Adams, V., and Askenazi, A., *Building Better Products with FEA*, On word Press, 1999.
3. Mott, L.M., *Applied Strength of Materials*, Prentice-Hall, 1996.
4. Buchanan, G.R., *Finite Element Analysis: Schaum's Outline Series*, McGraw-Hill, 1995.

End of Exercise

NOTES: