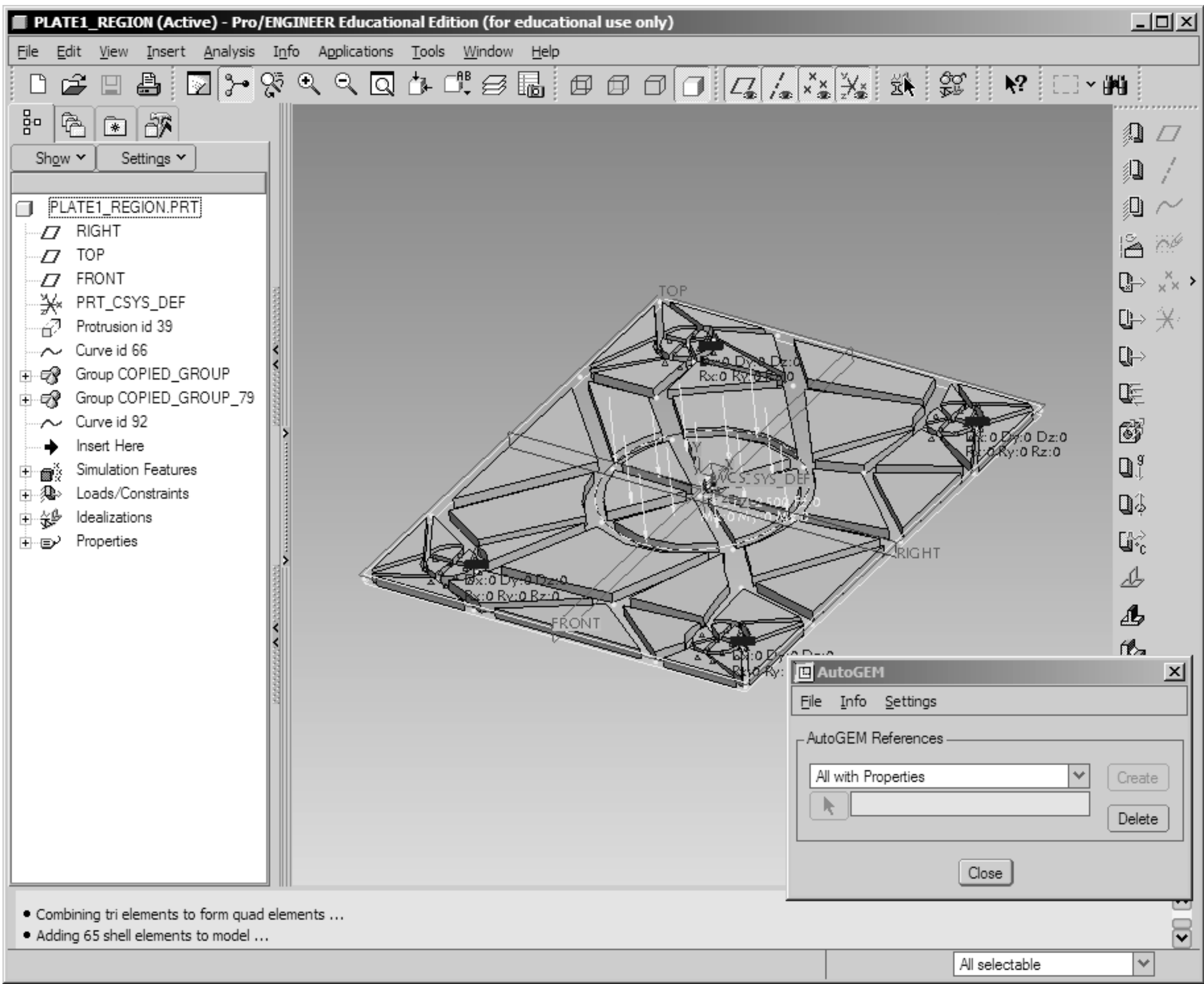
# Pro MECHANICA STRUCTURE WILDFIRE 4

## ELEMENTS AND APPLICATIONS – Part I




**Yves Gagnon, M.A.Sc.**
**Finite Element Analyst & Structural Consultant**

**SDC**
**PUBLICATIONS**

Schroff Development Corporation

**www.schroff.com**

***Estimated time: 1½ hours***

### 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 the following:

- 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®:

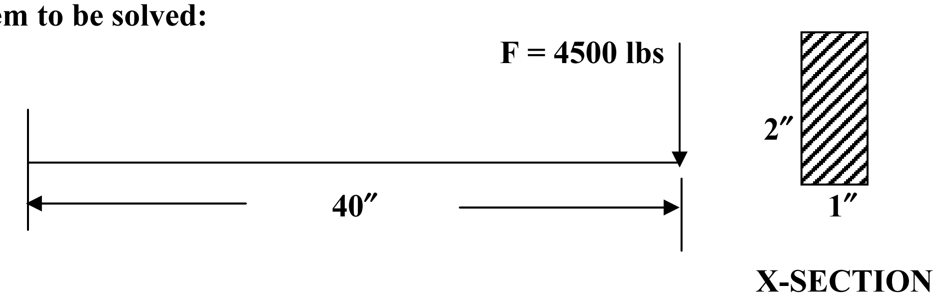
- The x-axis is along the length of the beam (the beam's axis);
- The y-axis defines the *perpendicular* orientation of the beam section with respect to the beam's axis;
- The z-axis will follow according the first two axis' above.

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 is fairly representative of a beam and that the aspect ratio (l/t) > 10.

## Procedure

(NOTE: Before you start, create a folder named "beam" using Explorer; this will be the working directory for the exercise.)

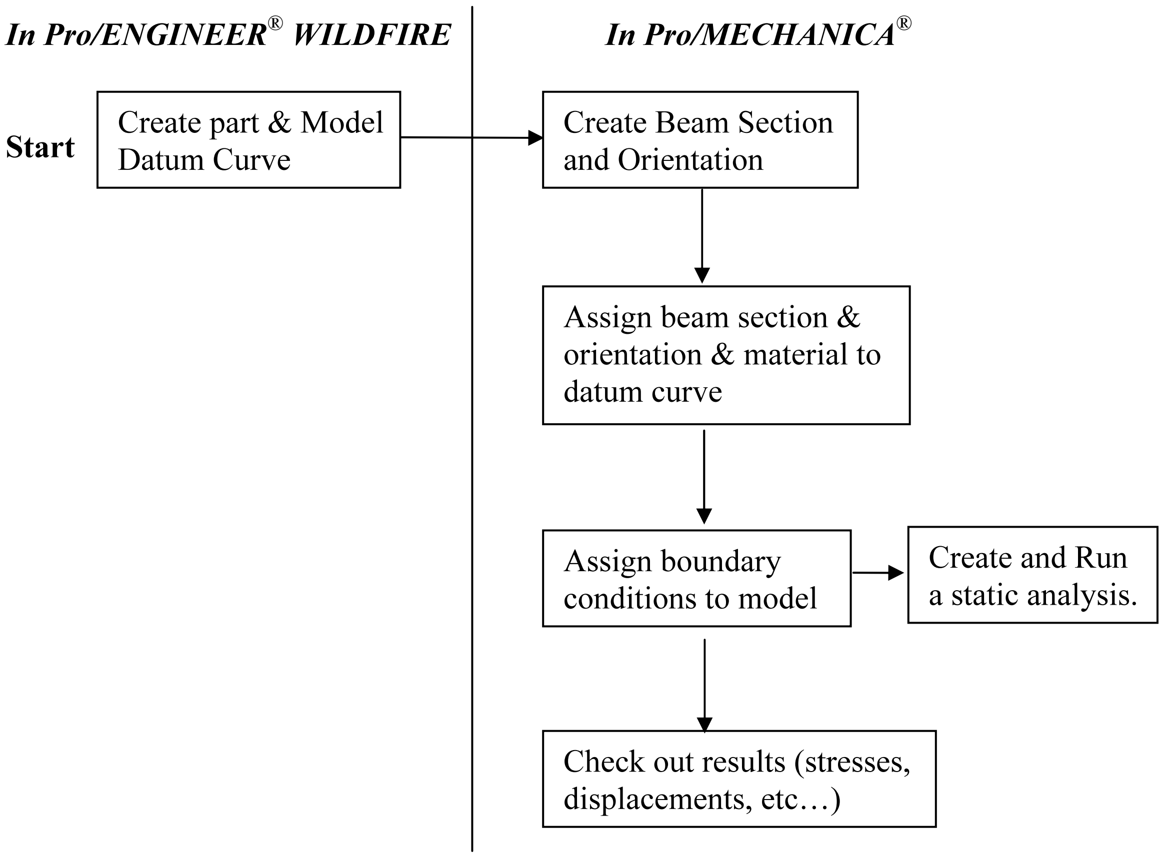
**The problem to be solved:**




Objectives: Draw Shear and Moment diagrams, and obtain the Maximum VM stress and maximum deflection fringe plots.

### 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 Pro/ENGINEER® WILDFIRE

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

**Start > Programs > PTC > Pro/ENGINEER**
(It takes approximately 30 seconds for Pro/ENGINEER® to start.)

Set up your working directory (**File > Set Working Directory**).


Figure 1

Select your working directory, then select **OK**.

## 2. Create a part named: beam

Select **File > New**.

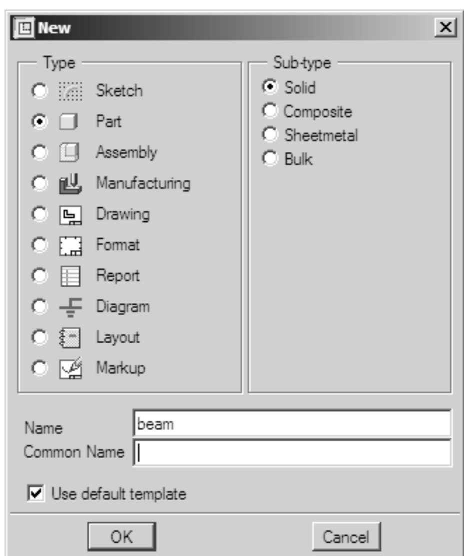



Figure 2

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

# 3. Setting up in Pro/ENGINEER® WILDFIRE for FEA Modeling

Before we go to MECHANICA®, we must understand how beam elements are created. Think of beam modeling as a sweep-type protrusion in Pro/ENGINEER® WILDFIRE. In the case of FEA modeling, a datum curve is created in Pro/ENGINEER®, then the beam cross-section (the beam element) is assigned in MECHANICA®.

## 3.1 Creating a Datum Curve for Beam Assignment

Click on the **sketch datum curve** icon. Use ***front as your sketching plane*** and accept the default viewing direction and other references. (See dialog box below.)

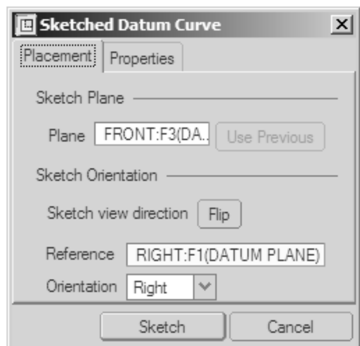
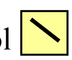



Figure 3

Click on **Sketch**. 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.

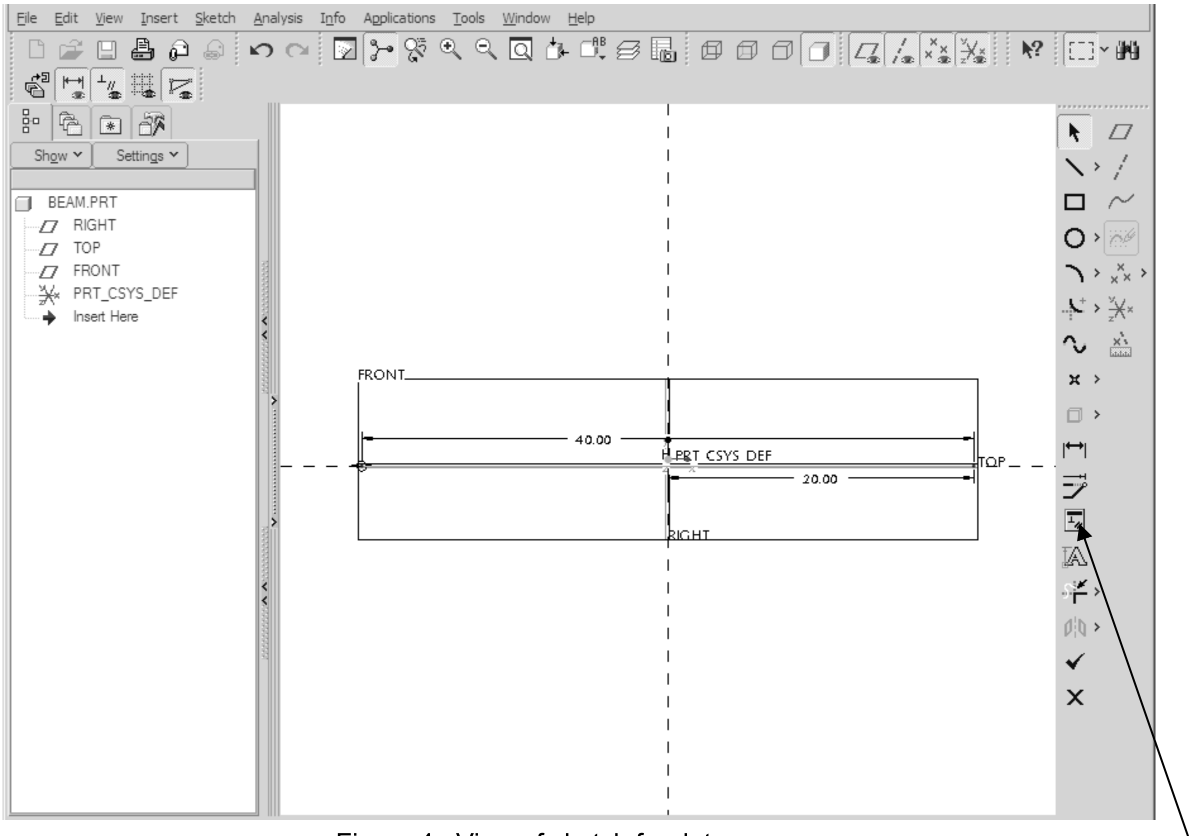



Figure 4: View of sketch for datum curve

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). Turn datum planes off using the datum icon at the top of your screen. Select **Tools** > **Environment** (pull-down menu) and *check off* the spin center for increased visibility. **Select OK**.

## Procedure in Pro/MECHANICA®

### 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* dialog box come up on the screen as shown in Figure 5.

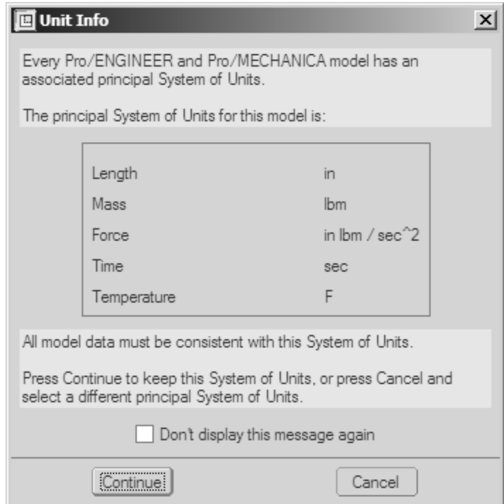



Figure 5: Unit Info Window

**Important:** Have a look at the working units and then Click ***Continue***. This box will only reappear for that part if Pro/ENGINEER® WILDFIRE is shut down. The following box will come up:

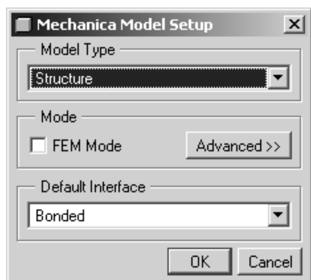



Figure 6

Select **Structure** from the menu, then **OK**.

# 2. Beam Assignment

## 2.1 Beam Section

Select **Insert > Beam**. The dialog box in Figure 7 will come up.

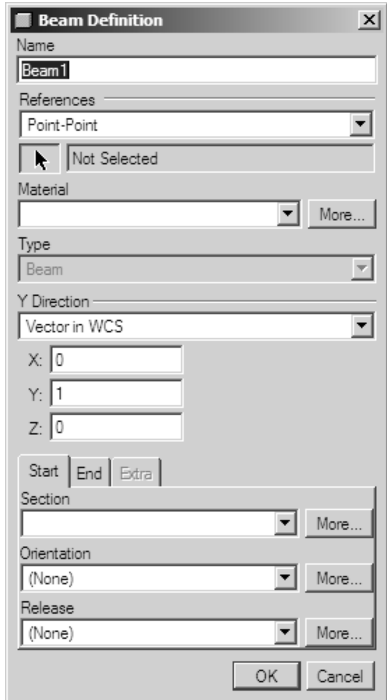



Figure 7

Under the **Start** tab, for the **Section** item, click on **More**; the following window will come up:

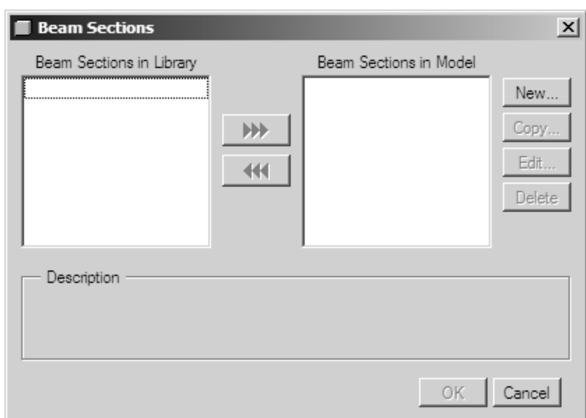



Figure 8

Click on **New**. The dialog box shown below will come up. Enter the information as seen in Figure 9.

**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 a rectangle 1 in wide and 2 inches high.*

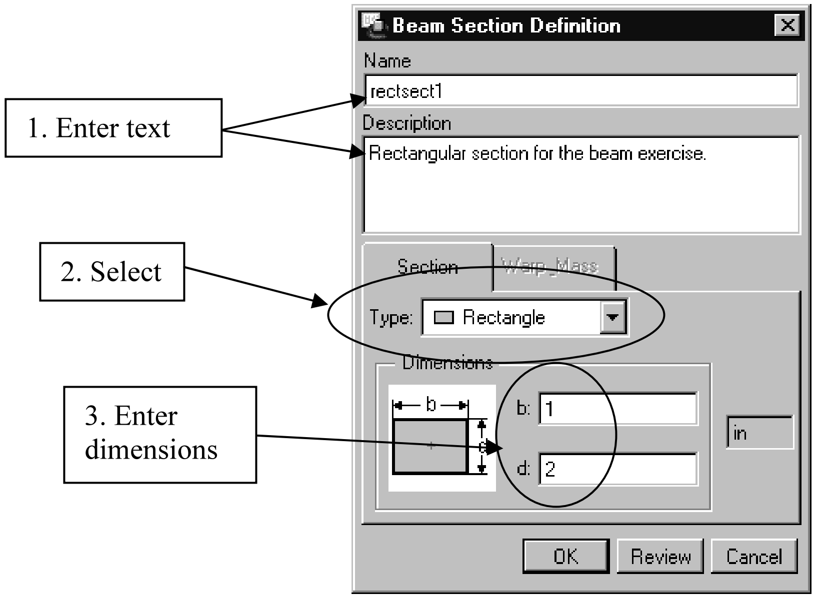



Figure 9

*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.*

Once the information is entered, click on **OK**. Then **OK** the beam section dialog box to return to the Beam Definition dialog box.

We must now orient the vector of the cross-section. Even though it will show up on the screen if we don't, Pro/MECHANICA® 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

Under the **Start** tab, for the **orientation** item, click on **More**. The Beam Orientation dialog box comes up; click on **New**. Enter the following information in the box as shown in Figure 10.

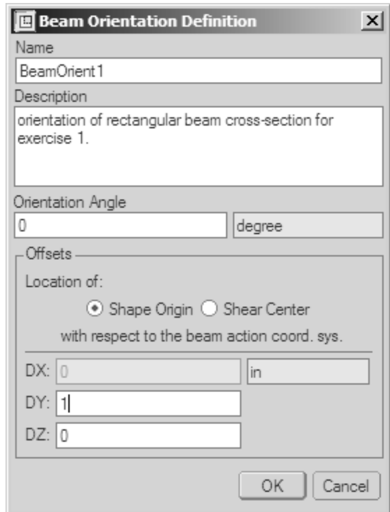



Figure 10

Defined by: Y direction of **vector in WCS** Vector (0, 1, 0). (This defines the orientation of the cross-section, as explained on the previous page in the *STOP* note.) Click on **OK** then **OK** for the Beam Orientation dialog box.

## 2.3 Beam Definition

Complete the Beam Definition window as follows (also see the next page for menu selection description):

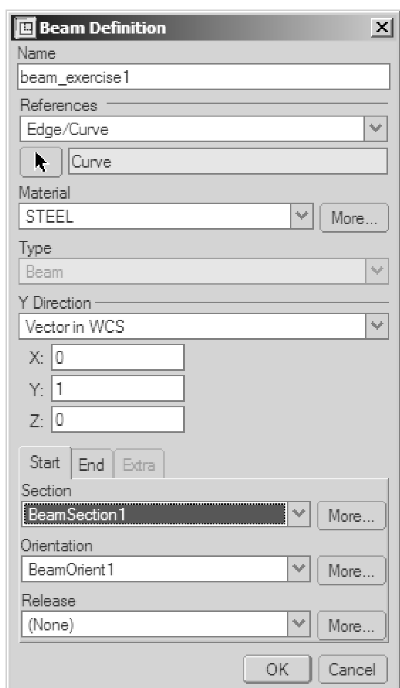



Figure 11

Procedure to complete Figure 11:

- Under Name: enter **beam_exercise1**.
- Under References: select **Edge/Curve**. **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 **More…** and the following window will come up:

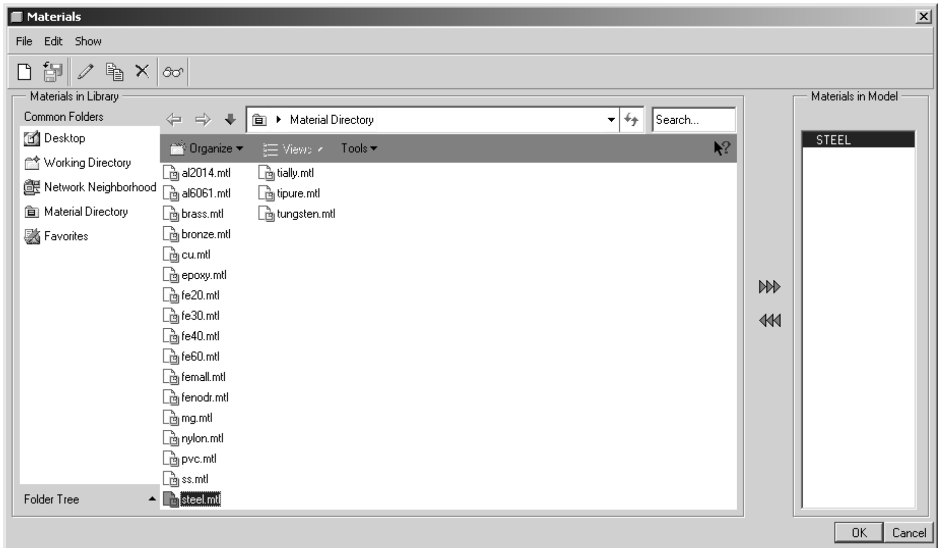



Figure 12

Select **Steel** by **scrolling down the left column** and **moving it to the right column** by clicking on the triple arrows box in the center. Then click on **OK**. There are no beam releases for this analysis. Select **OK** and note that ***small rectangles*** appear along the datum curve. Zoom in on one of the rectangles and look!

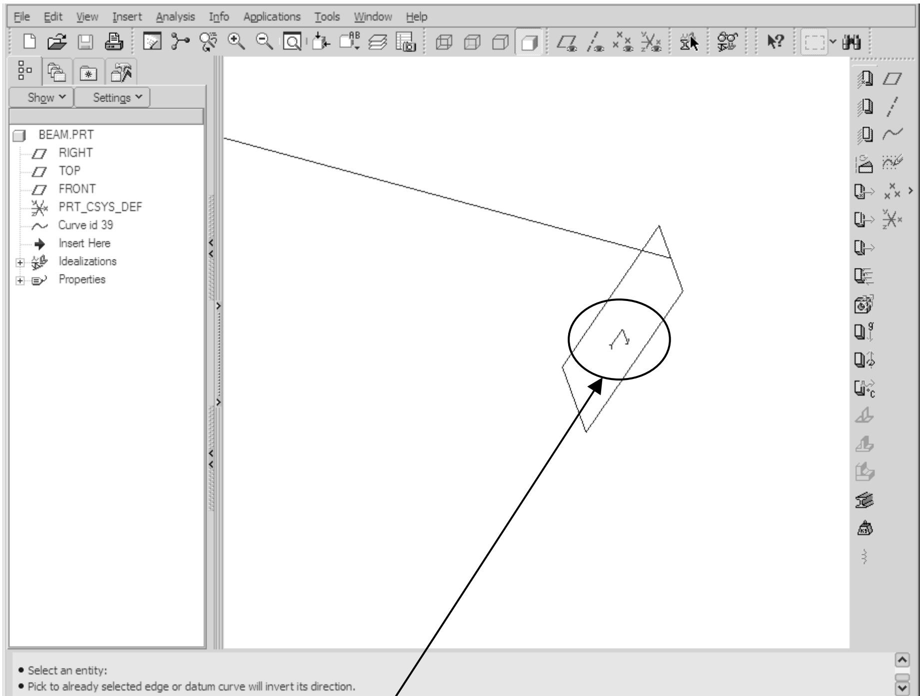
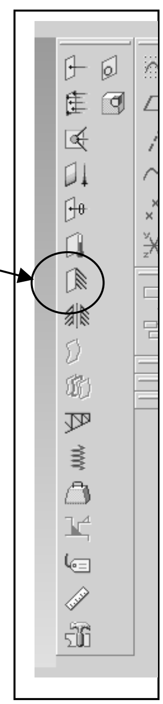



Figure 13

You should be able to see a vector inside the beam. This vector is oriented in the coordinates given above (0, 1, 0). You could easily change the orientation of the rectangular section and the vector simply by changing the directional coordinates of the vector. Our beam idealization is fully defined. Good job!

## 3. Constraints



Select **Insert > Displacement Constraints** or click on the icon as shown to the right.

The following dialog box will come up. Enter and select the information as shown:

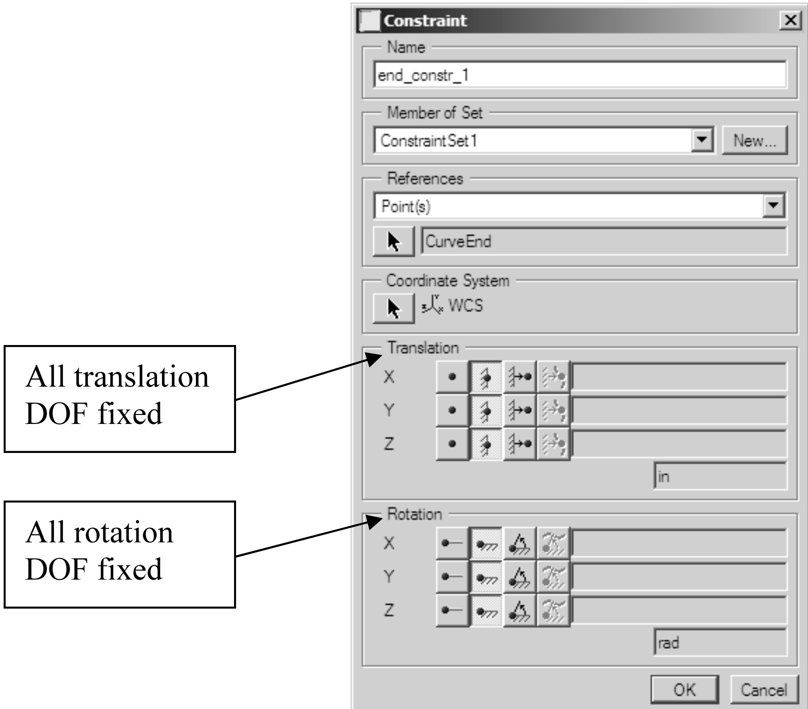
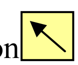



Figure 14

- ➢ Name: end_constr_1
- ➢ Member of set: ConstraintSet1
- ➢ Under References, **select Points**. Click on the arrow icon  and select the end point as shown in Figure 15:

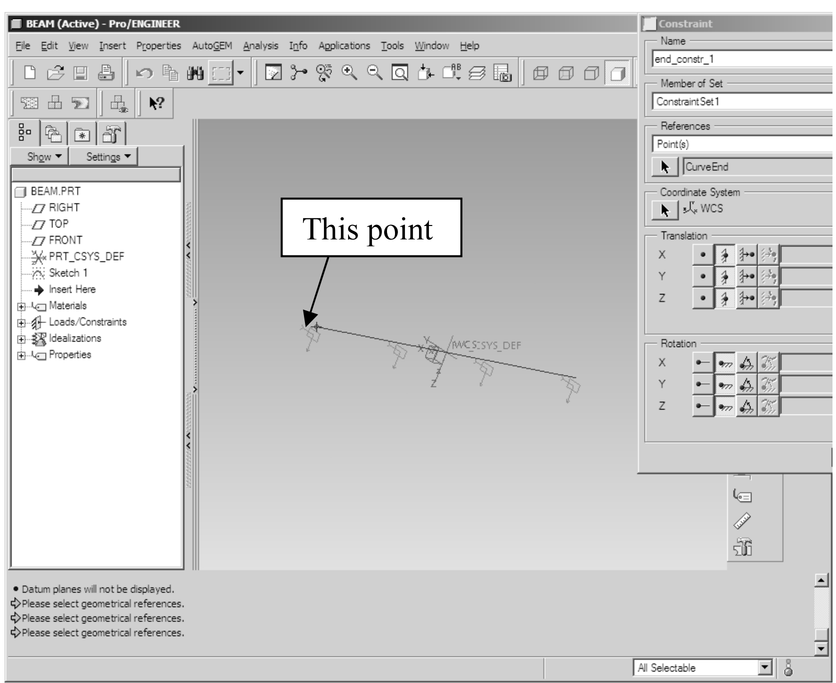
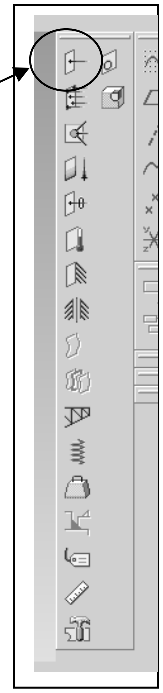



Figure 15

**Select OK** to close the Constraint Definition box.



## 4. Loads

Select **Insert > Force/Moment Load**
or click on the icon as shown to the right.

The following dialog box will appear. Fill out the information as seen in Figure 16.

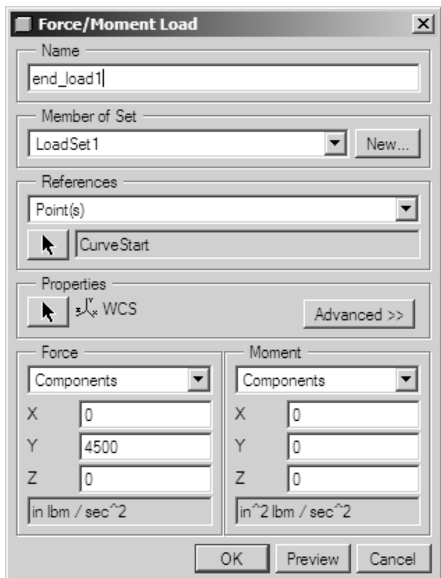
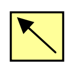



Figure 16

Under References, select **Points**. Click on the arrow icon  and select the end point as shown below:

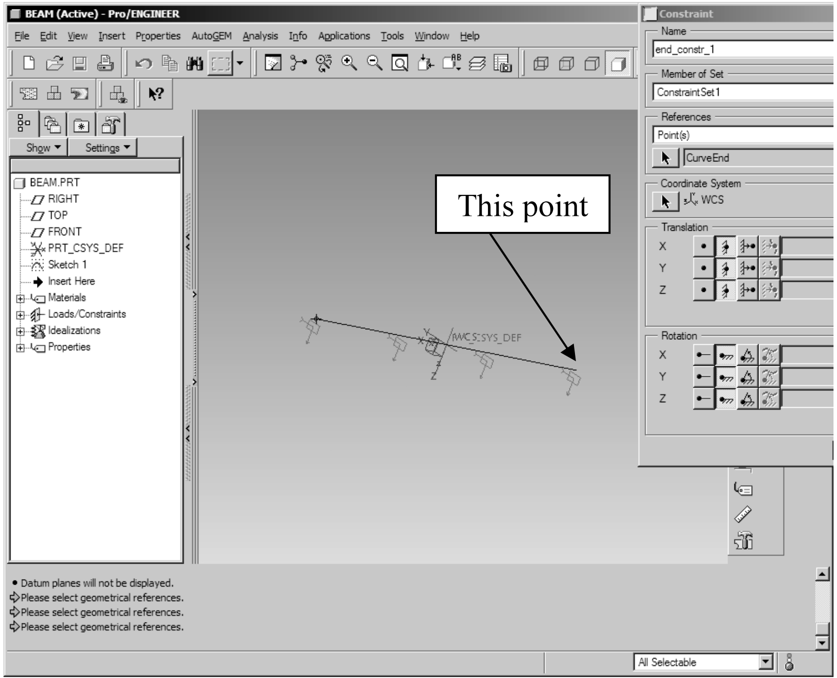



Figure 17

Similar to the constraint point selection, select the **vertex** (located on the **opposite end** of the beam. Click **OK**. Under **Force > Components**, enter 4500 in the Y direction box.

Select **OK** off the Force/Moment dialog box. In order to properly show the load, from the *top pull-down menu* select **View > Simulation display** and select appropriate selections as shown in the dialog box below:

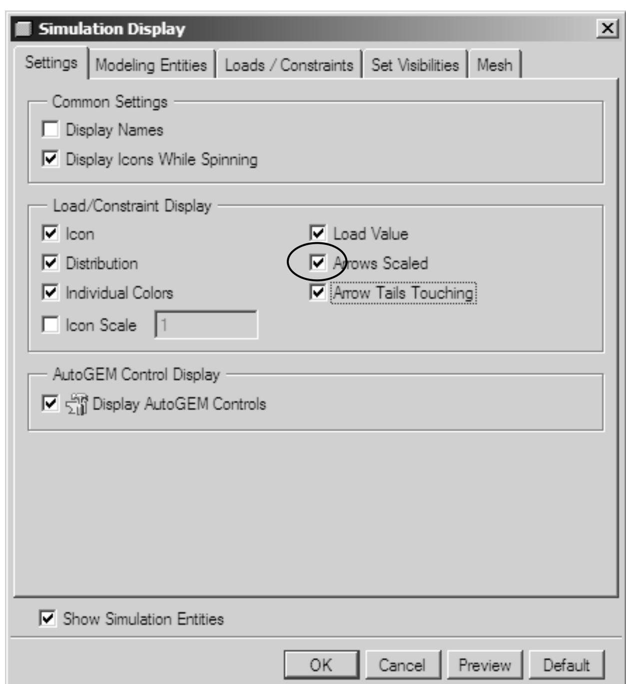



Figure 18

Click on **OK** to close the window. The resulting window should look like the following (default view):

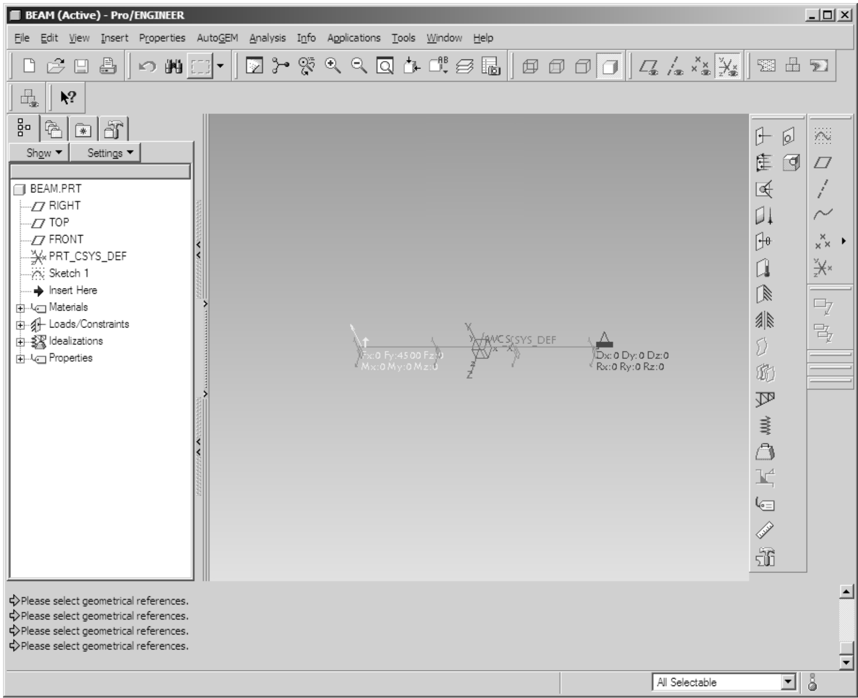



Figure 19

## 5. Generating Beam Elements

We now need to generate the finite element mesh. The nice feature about Pro/ENGINEER® WILDFIRE is that now we can control the mesh of the model without having to go to independent mode. There will be more applications of this new feature in subsequent exercises. For the time being, Select **AutoGEM > Create.** The following box will come up:

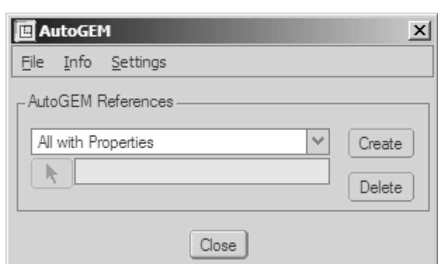



Figure 20

This box controls the mesh features, type and density. Click on **create.** The following dialog box will come up. (If the Create button is grayed out, review that the elements and material have been set properly)

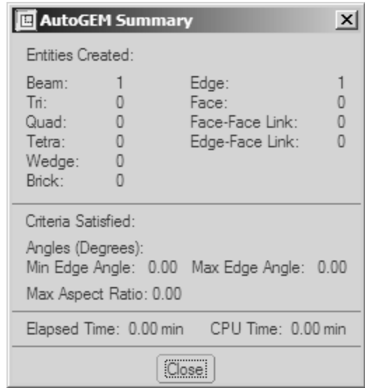



Figure 21

One beam element has been created. Click on **Close > Close** and answer **Yes** to the following:

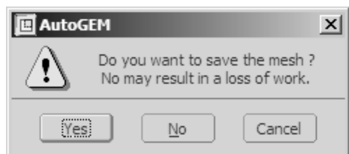



Figure 22

# 6. Set up and Run Analysis

## 6.1 Static Analysis

We will now attempt the static analysis of the beam. Select **Analysis > Mechanica Analyses-Studies** from the right menu; the window shown in Figure 23 will open.


Figure 23

Select **File > New static** from the top left corner menu. The following dialog 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 10 percent convergence.
- Converge on local displacement and strain energy and global RMS stress.

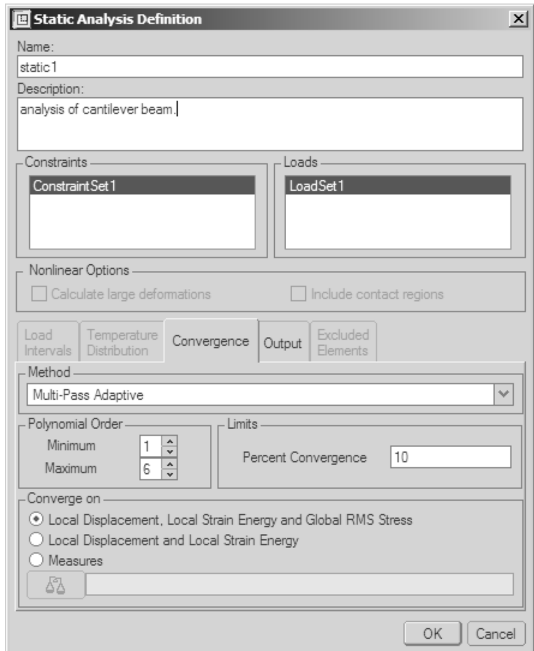



Figure 24

Click on **OK**.

### 6.2 Running the Analysis

With the following window still up, select the analysis, use your right mouse button to scroll down to **Start**. The analysis will now be running. The software will ask you if you want error detection and see the diagnostics window. Answer **Yes**.

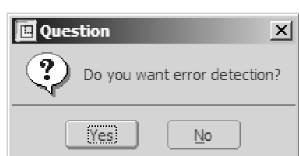



Figure 25

To check things as the calculations are performed, still using the right mouse button, select **Status**. (Note how fast the solution converged). From the status summary file, the following information is important to consider.

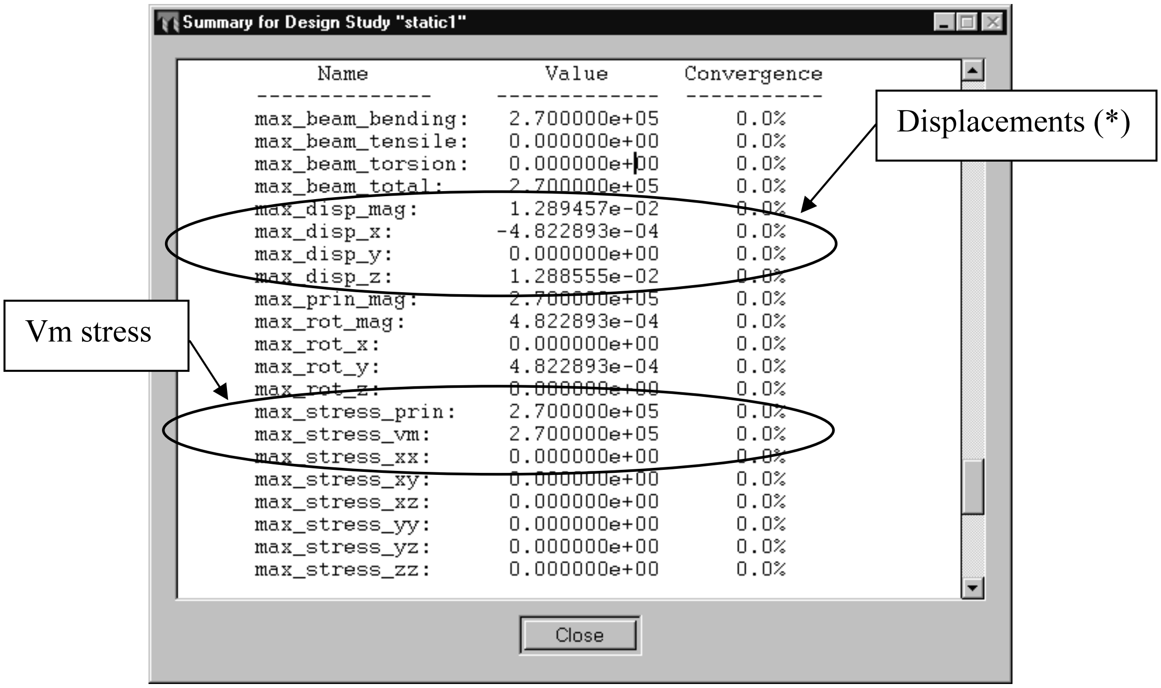



Figure 26

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 from Mechanics of Solids. The following table gives an overview of what details you should look for from the results of an analysis:

TABLE 1.1: OVERVIEW OF SUMMARY FILE

| 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 **Status** under the **Info** 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 **Status** under the **Info** 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. Select **Close** and **Close**.

This will bring you back to the FEA model with the MEC STRUCT menu on the right hand side.

**Close** the status window if not already done so.

## 7. Analysis of Results

Select **Analysis** > **Results**. Once the empty window comes up, **click** on this icon named '**insert a new definition**.'


Figure 27

The following box will come up.

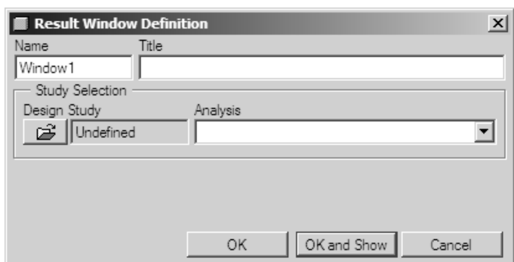



Figure 28

Complete the result window definition as shown in figure 29. Under name, type in **vm_stress** (for Von Mises stress).

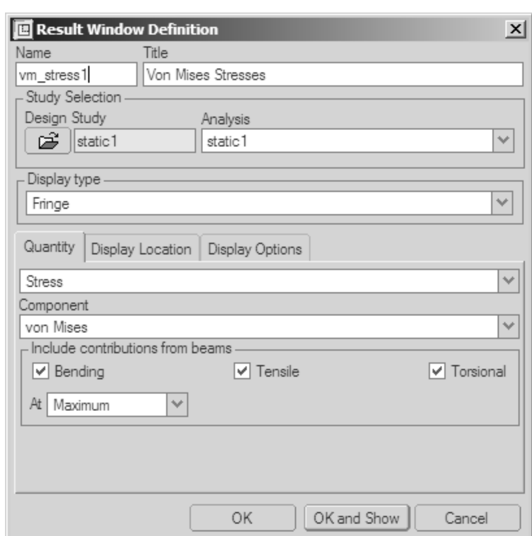



Figure 29

Select the open file folder and select **static1** for the analysis (click **Open**). Then fill out the contents definition box as shown above. Once completed, select **OK**. Then from the pull-down menu, select **view > display**. The following window will come up:

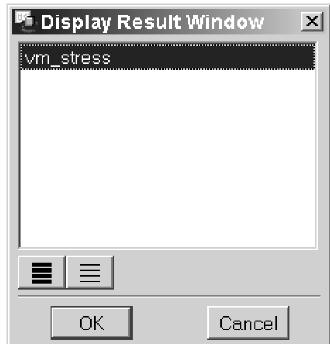



Figure 30

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

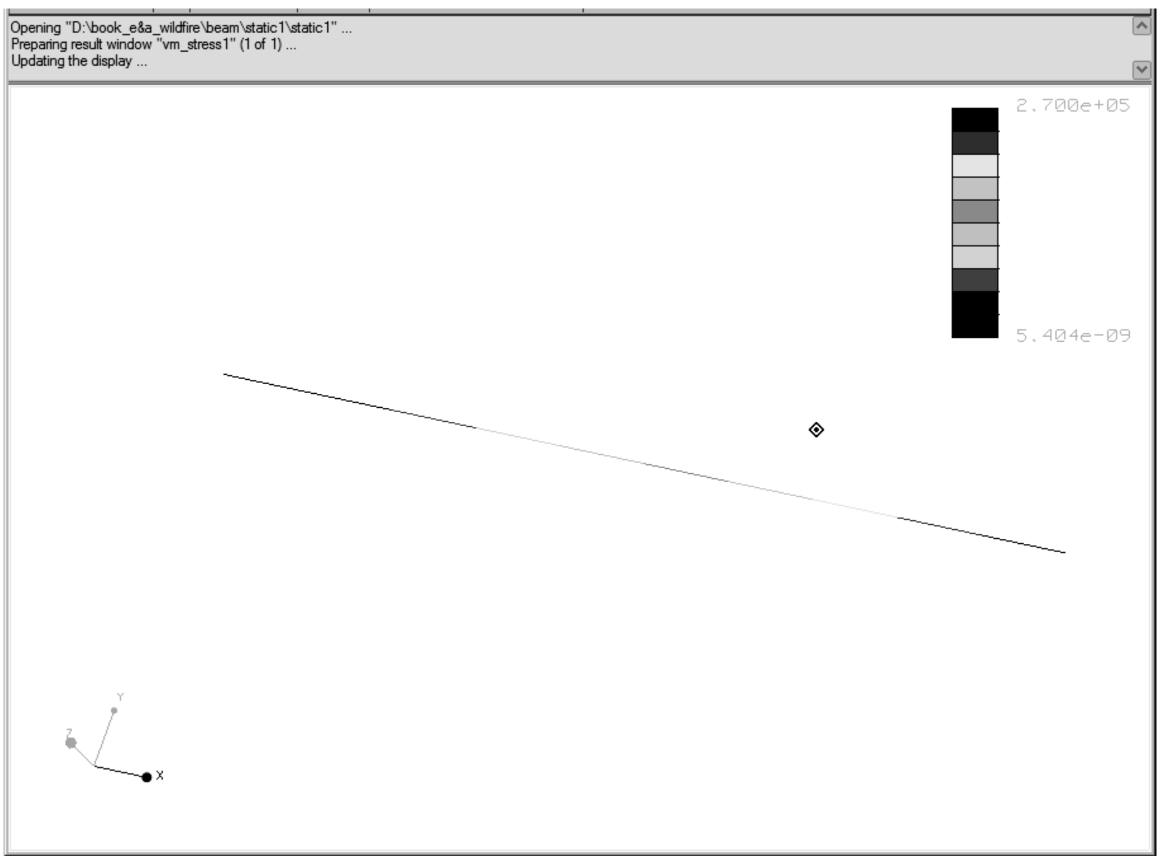



Figure 31: Von Mises Stress Results for Beam Analysis
(difficult to reproduce on a document)

The results from a beam analysis are not the most impressive to look at; and are very difficult to reproduce on a document. But they provide accuracy, given that your geometry and boundary conditions are precise. The only colors that we see are along the beam curve that represents the point on the beam cross-section where the Von Mises maximum stress is located. 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 a practice exercise, create the following result window in order to look at the maximum displacement from the model. Simply follow the procedure in item 7 above. The only changes being under quantity on the definition form.

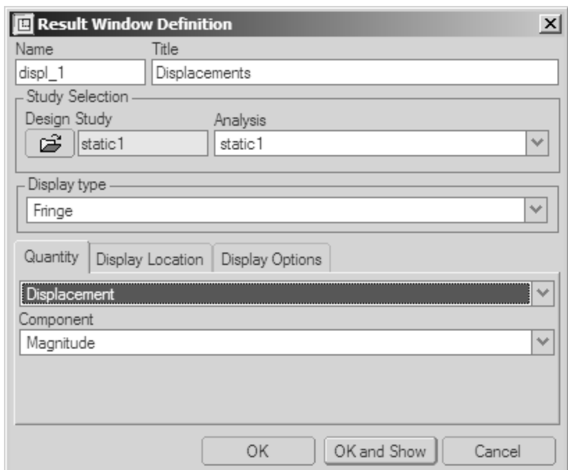



Figure 32

Select **OK and Show**.

## Approximate Finite Element Analysis Solution (Theory Warning)

**Don't skip this! There are some more Mechanica® procedures included with it to make it more fun.**

In solving the beam analysis using FEA theory, the problem can be seen as follows, including all reactions and forces:

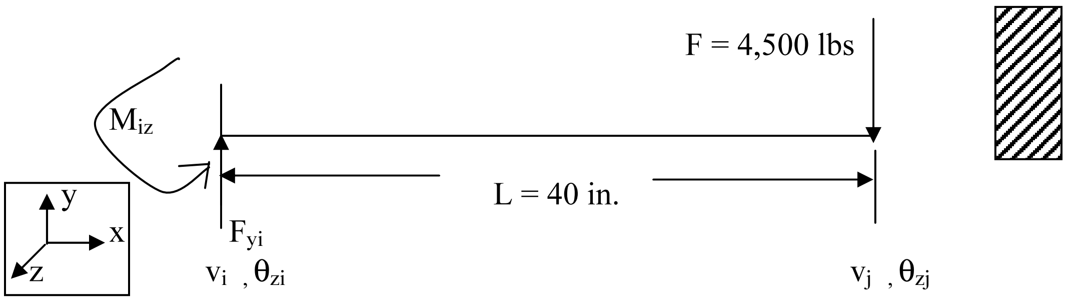



Figure 33

Table 1.2 gives the symbol definitions:

TABLE 1.2: SYMBOL DEFINITIONS

| Symbol | Definition |
|---|---|
| $L$ | Length of beam. |
| $M_{iz}$ | Bending moment at the fixed end of the beam. |
| $F_{yi}$ | Vertical reaction at the fixed end of the beam. |
| $E$ | E is the young's modulus of elasticity of the material. |
| $I$ | The 2nd Moment of Area. |
| $d^2v/dx^2$ | The curvature of the beam (not shown). |
| $v_i$ | The vertical deflection at x = 0. |
| $\theta_{zi}$ | The beam slope at x = 0. |
| $v_j$ | The vertical deflection at x = L. |
| $\theta_{zj}$ | The beam slope at x = L. |

The general equation for beam deflection is from beam theory as follows:

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

*where*;

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

Consequently:

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

Let’s get the moment diagrams from Mechanica®; this will be fun. From your beam model, where you are in **Applications > Mechanica > Structure**, select **Results** > then **click** on the ‘**insert a new definition**’ icon.


Figure 34

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 in Figure 35.


Figure 35

Enter and select the following information as indicated in the figure above:

- **Name and Title**: V_M and Shear and Moment Diagrams
- **Design Study**: Select the open folder and open the static1 analysis.
- **Display Type**: Select Graph.
- **Quantity:** Select shear and moment (and de-select P, Vz, Mx and My as they are not necessary for this two-dimensional 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.)
- **Graph Location:** Make sure that **curve** is selected. **Click on Select and select the beam curve off the screen. Click on the middle mouse button.** You will then be prompted with the following message as MECHANICA® highlights one of the ends of the beam. Select **toggle** once. Then **OK**.


Figure 36

Click on **OK**, then **Accept** and **Show**.

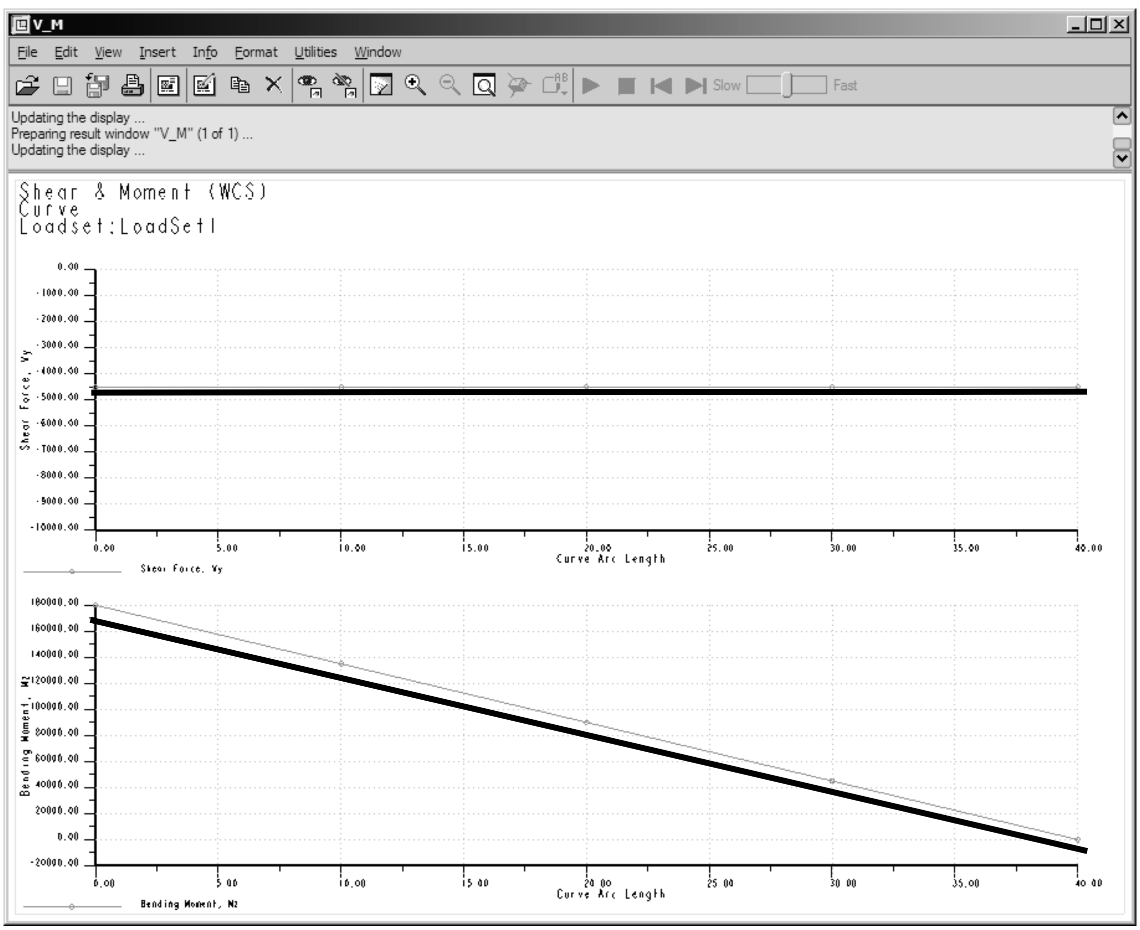



Figure 37: Shear and moment diagrams

It may be difficult to see the numbers on the graphs above but on your screen, you should get the results we were expecting, where the maximum shear is 4,500 lbs and the maximum bending moment is 180,000 in.lbs. Select **File > Exit Results**. Do not save the result window.

Now let's integrate twice equation 1-1 above. This will give the following:

$$EI_z \frac{dv}{dx} = \frac{F_{yi} x^2}{2} + M_{zi} x + A1 \qquad \boxed{\text{1}^{\text{st}} \text{ integration}}$$

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

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

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

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):

$$M_i = \frac{6EI_z}{l^2} v_j + \frac{2EI_z}{l} \theta_{zj}$$

$$F_{yi} = \frac{12EI_z}{l^3} v_j + \frac{6EI_z}{l^2} \theta_{zj} \qquad (1\text{-}2)$$

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 X $10^6$ 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):

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

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

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

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*

$$\left(\begin{array}{cc|c} 1 & l/2 & \frac{l^3}{EI}375 \\ 0 & 1 & -\frac{l}{EI}180000 + \frac{l^2}{EI}2{,}250 \end{array}\right)$$

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

**θzj = -.186 rd** and
**$v_j$ = 4.95 in.**

Comparing the deflections:

| **FEA Approximate Solution** | **Pro/MECHANICA® Solution** | **Exact Solution ($PL^3$ / 3EI)** |
|---|---|---|
| 4.95 in. | $1.289 \times 10^{-2}$ ($sec^2$) | 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 4.96, almost the magnitude of the exact solution.

## Conclusion

Beam elements are fast and accurate elements. They are mainly used by analyst 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, so 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).

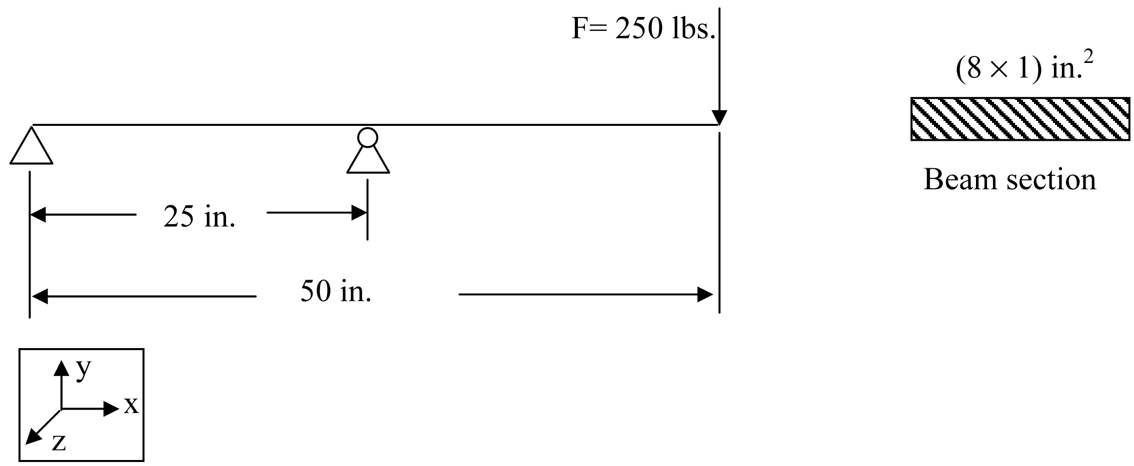
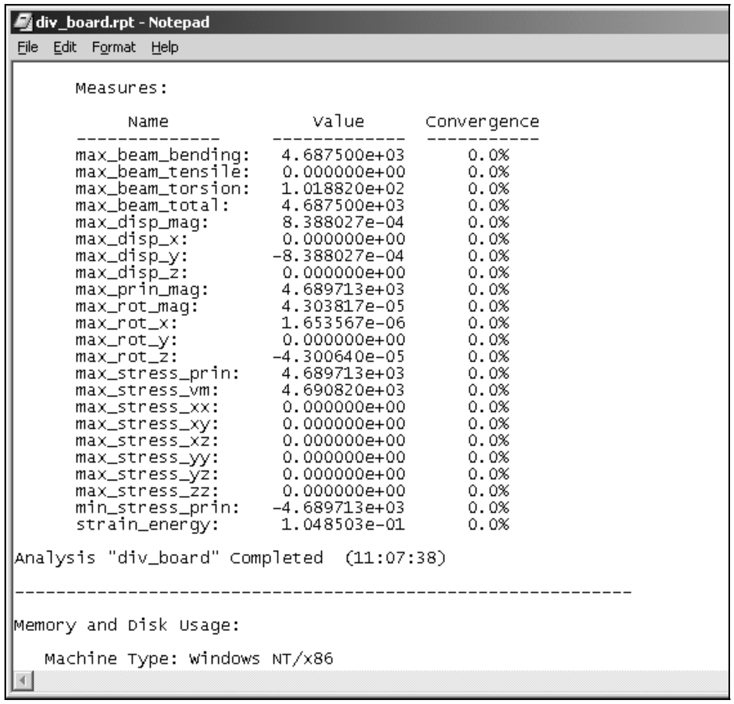



Figure 38: Diving Board Loading Diagram

Notes and Hints:

1. △ Fixes the location in all 6 DOFs.
2. Fixes the y direction DOF and the rotation with respect to x axis.
3. Material: Use Aluminum 2014.
4. Create a datum point at the center reaction, use vertex for the left reaction and the load.

## Results



div_board.rpt - Notepad
File Edit Format Help

```
     Measures:

          Name              Value       Convergence
     --------------     -------------   -----------
     max_beam_bending:    4.687500e+03      0.0%
     max_beam_tensile:    0.000000e+00      0.0%
     max_beam_torsion:    1.018820e+02      0.0%
     max_beam_total:      4.687500e+03      0.0%
     max_disp_mag:        8.388027e-04      0.0%
     max_disp_x:          0.000000e+00      0.0%
     max_disp_y:         -8.388027e-04      0.0%
     max_disp_z:          0.000000e+00      0.0%
     max_prin_mag:        4.689713e+03      0.0%
     max_rot_mag:         4.303817e-05      0.0%
     max_rot_x:           1.653567e-06      0.0%
     max_rot_y:           0.000000e+00      0.0%
     max_rot_z:          -4.300640e-05      0.0%
     max_stress_prin:     4.689713e+03      0.0%
     max_stress_vm:       4.690820e+03      0.0%
     max_stress_xx:       0.000000e+00      0.0%
     max_stress_xy:       0.000000e+00      0.0%
     max_stress_xz:       0.000000e+00      0.0%
     max_stress_yy:       0.000000e+00      0.0%
     max_stress_yz:       0.000000e+00      0.0%
     max_stress_zz:       0.000000e+00      0.0%
     min_stress_prin:    -4.689713e+03      0.0%
     strain_energy:       1.048503e-01      0.0%

Analysis "div_board" Completed  (11:07:38)

------------------------------------------------------------

Memory and Disk Usage:

   Machine Type: Windows NT/x86
```

Figure 39: Summary file results for Project 1

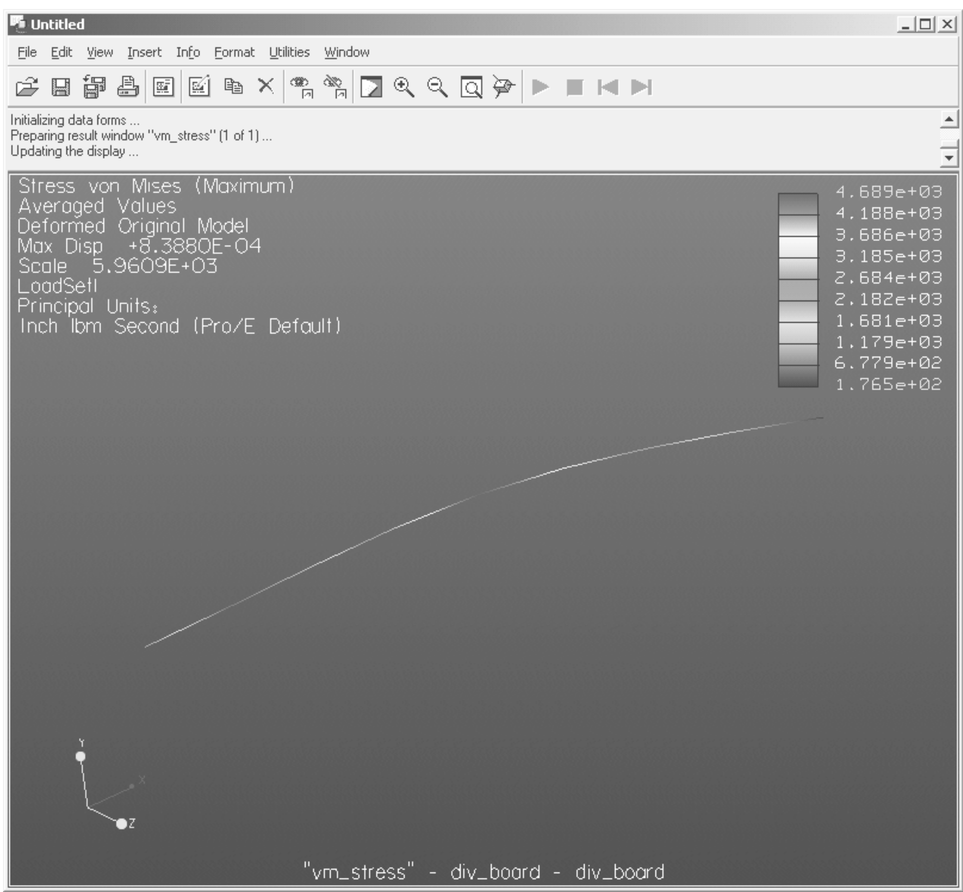



Figure 40: Von Mises Stress fringe plot for Project 1

## References

1. Toogood, Roger, ***Pro/MECHANICA Structure Tutorial (release 2000i)***, SDC Publications., 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**