PRO-MECHANICA

Pro-Mechanica is a product of PTC. It works with Creo Pro/E in integrated mode to allow users to perform structural and thermal analyses. This tutorial was originally written for UNIX platform, but updated to be valid for the Windows version.

Lesson One < Structural > Beam

Cantilever Beam

Given: A cantilever beam with length L and a rectangular cross section with dimensions as shown in Figure 1. The beam is subjected to concentrated load F applied at the free end.

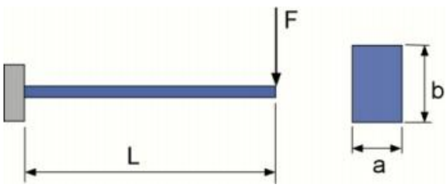


Figure 1.

F = 100,000 N, L = 2 m, a = 0.050 m, b = 0.100 m,

Find:
(1) Shear diagram, moment diagrams, maximum deflection, and maximum von Mises stress using Pro/Mechanica.
(2) Deflection of the beam using exact solution approach and FEM approximation solution approach and compare the results.

Solution:

The general approach in Pro/M is as follows:

I- Create the model in Pro/E

II- Switch over to Pro/M integrated mode then

- Assign material
- Define geometric constraints
- Apply load constraints
- Define the type of analysis

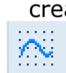
III- Run the analysis

IV- Post-processing: Study the results and carryout verification.

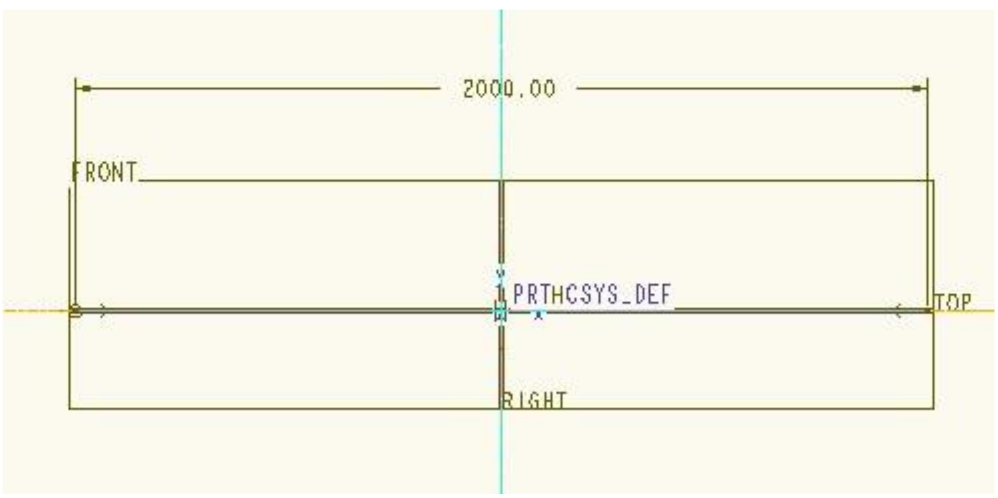
I. PRE-PROCESSING WITH PRO/E

STEP 1: Create a file name "beam" by clicking on the icon or alternately, CTRL+N. A New File dialog window will pop up. Enter "beam" and leave default settings of part type and solid sub-type.

STEP2: Set unit to mmNs setting by choosing **File > Properties > Units > Change** from Part window. Select mmNs option followed by clicking on the SET button. Accept the "Convert Existing Numbers (Same Size)" option from the Warning window. Click OKAY and close the Unit Manager window.

STEP 3: Establish the beam by creating a curve with the sketch tool. Click on the sketch tool curve creation icon . From CRV OPTIONS window, select **Sketch -> Done**. For sketching plane, use the FRONT plane. Accept the default sketch orientation by clicking Sketch (Reference: RIGHT, Orientation: Right).

STEP 4: In the sketcher mode, draw a line by selecting the line icon and draw a line aligned with the TOP plane and symmetrical about the RIGHT plane. To do this, create a centerline along the RIGHT plane and create a symmetric constraint between the two end points of the line and the centerline (This is just my convention. You may choose your own reference system for the sketch but you need to keep track of the coordinate system in which you define the line). Make change to the dimension to 2000 from end to end as shown below.

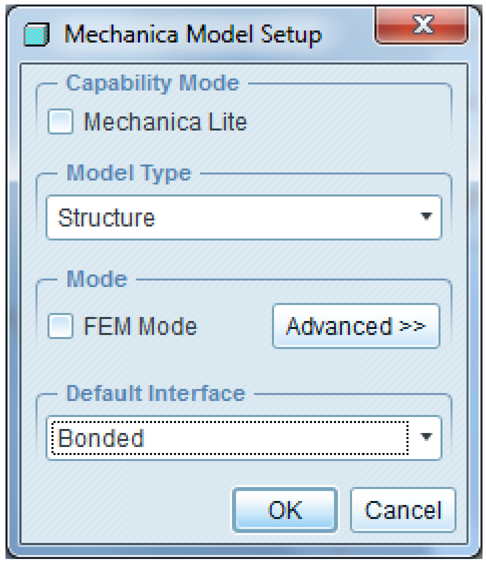


Once done, exit sketcher mode by clicking on the icon in the toolbar then click OKAY from the Reference dialog box to complete the section definition.

II. PARAMETER DEFINITION IN PRO/MECHANICA

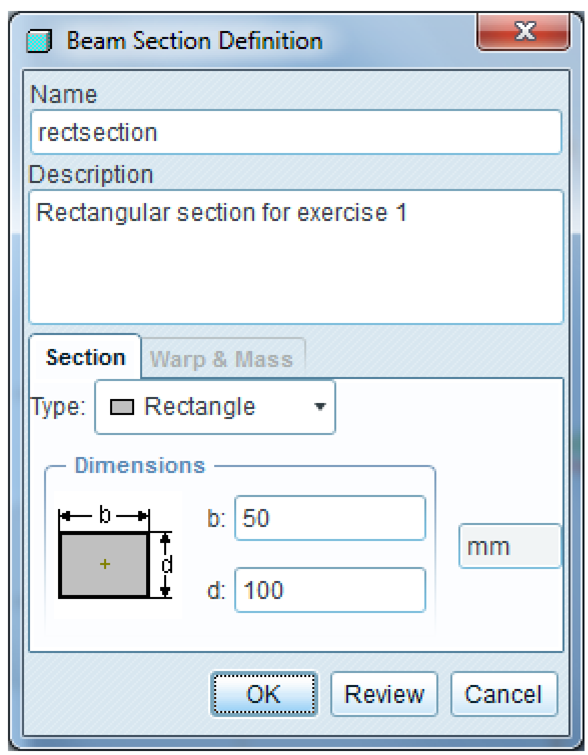
STEP 5: Entering Pro/M by selecting **Applications** -> **Mechanica** menu. A Unit Info window will pop up to verify the unit setting you have defined back in Pro/E. Make sure review the displayed setting and click on the check box to keep the display from poping again every time you are accessing Mechanica within the same session.

STEP 6: From Mechanica menu, select Structure submenu.



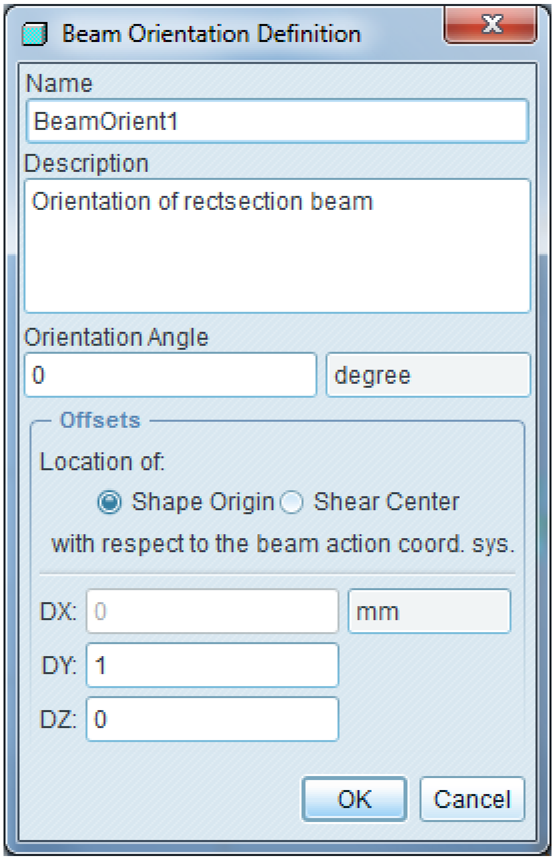
STEP 7: Defining beam's cross section. Use the **Properties>Beam Sections** command to define the shape and size of the cross sections when you create beams. You can also define a beam section and save it in a library file, called mbmsct.lib, for future use.

In the "Beam Sections" window, click on NEW button to enter the beam definition dialog box as shown below.



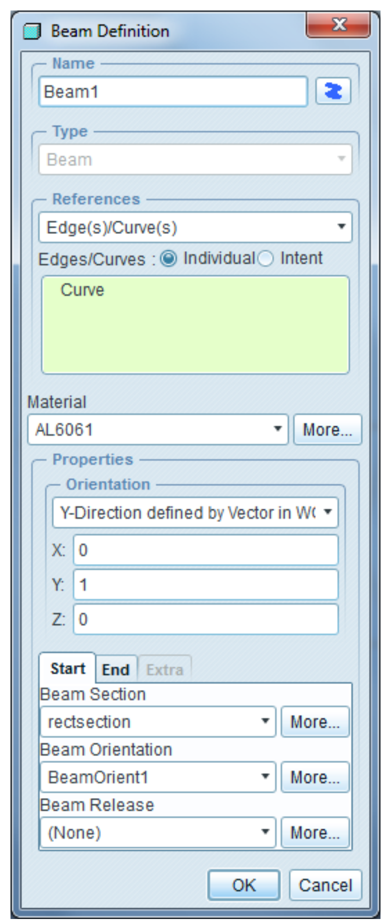
Enter "rectsection" in the Name prompt (you may choose whatever name you wish but you need to keep track of the names for reference later on in the session). Enter a description of your beam (optional). In the Section group, pull down the Type menu and choose Rectangle option as shown above. In the corresponding dimension box, enter "50" for b dimension and "100" of d dimension (note that the unit is mm). After the completion of dimension entries, click on **OK** followed by **Close**.

STEP 8: Defining beam's orientation. Now that the cross section has been defined, you must specify how the beam section is oriented with respect to a chosen coordinate system (WCS is default). Select **Properties** > **Beam Orientations** and click on **NEW** to enter the Beam Orientation Definition as shown in the screenshot below:



Click Ok and Close

Note that the vector [0,1,0] defines the orientation of the beam with respect to the beam action coordinate system. This vector will then be "aligned" with the WCS in the following step.



STEP 9: Beam definition. In this step the beam material and orientation with respect to the WCS is assigned/defined. **Insert>beam**. A Beam Definition dialog will pop up as show below.

Enter 'Beam1' for Name

Select Edge/Curve from References pull-down menu.

Click on the “Select geometrical references” box then select the datum curve you've created.

Click on  button and select Aluminum 6061 as the material to be assigned to the beam.

Make sure Beam is selected under Type (default)

Accept the default setting "Y-Direction defined by Vector in WCS" in the Y Direction choice. For X: 0; Y: 1; Z: 0.

Make sure "rectsection" (or whatever name you've chosen for the beam section earlier in the process) is selected in the Section pull-down menu. Also check on the orientation name to be "BeamOrient1" as you have defined earlier.

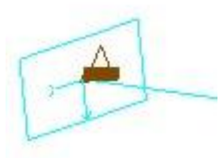
Leave [None] for Release

Click on **OK** button when done followed by **Done/Return** (if necessary). The rectangular sections will appear along the beam in red color.

STEP 10: Applying geometric constraints (boundary condition). Select **Insert** ->**Displacement Constraint**. A Constraint dialog panel will pop up. Enter the following:

Name: end_constraint
Member of Set: Constraintset1
References: Select References>Points, then click "Select geometrical references," then select the left (negative x-axis) end of the beam. Leave the WCS as is and accept all default constraints in both translation and rotation DOF's. This simulates the fixed-end condition of the beam.

Click on **OK** to complete the constraint definition. A constraint icon should appear next to the constrained point you've created as shown below:



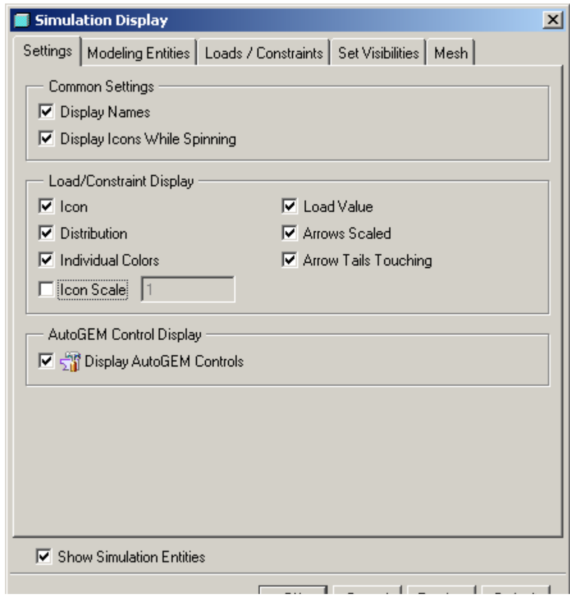
STEP 11: Applying loading condition. Select **Insert** -> **Force/Moment Load**.

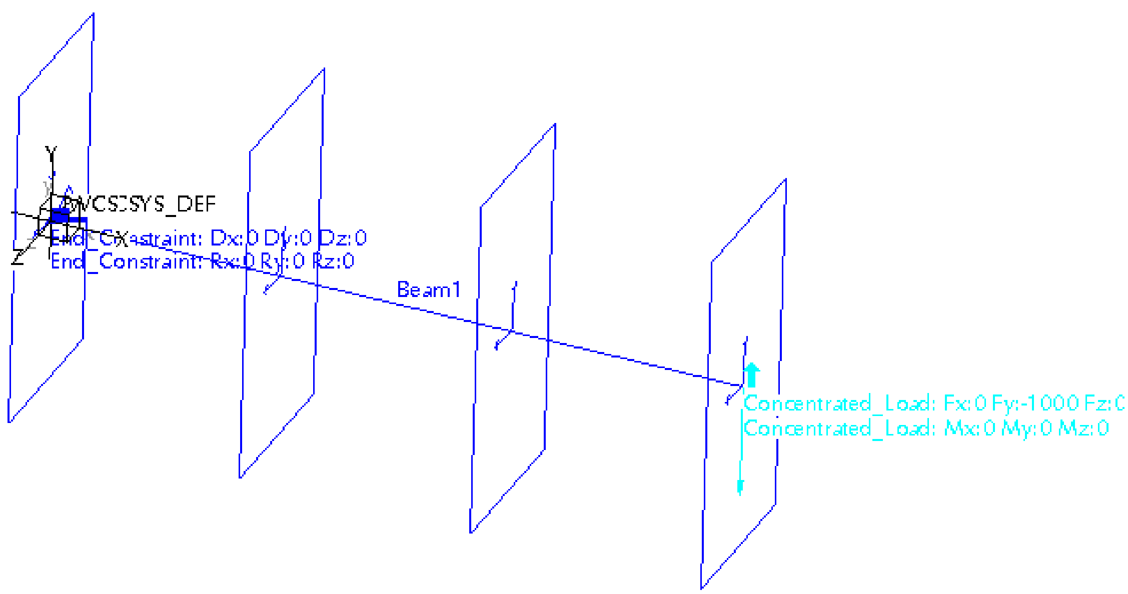
In the Force/Moment dialog panel, fill in the followings:

Name: concentrated_load
Member of Set: (leave default "Loadset1")
Select **References**>**Points** from the menu, click "Select geometrical references" then select the other end of the beam (the one that is not constrained).

Enter **"-100000**" in the Y-component box under Force group and leave all other zero. Click **OK** button (You may use the **Preview** button to see the applied load).

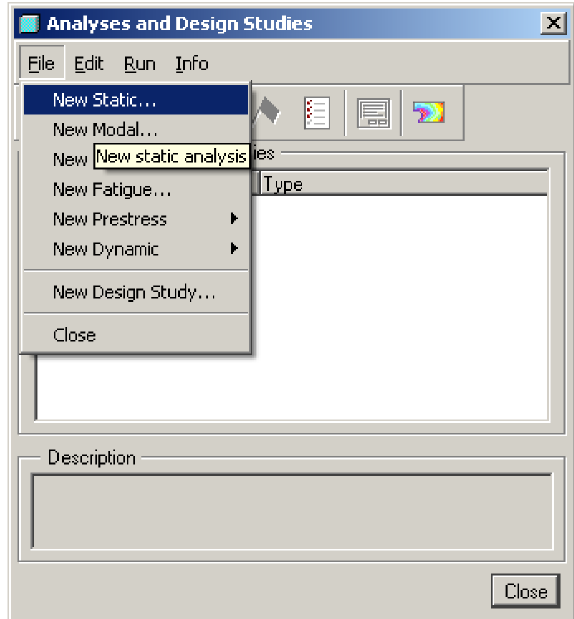
You may also show the applied along with other constraints by selecting **View** -> **Simulation display** to bring up the dialog panel as shown below (optional).


Simulation Display
Settings
Modeling Entities
Loads / Constraints
Set Visibilities
Mesh
Common Settings
Display Names
Display Icons While Spinning
Load/Constraint Display
Icon
Load Value
Distribution
Arrows Scaled
Individual Colors
Arrow Tails Touching
Icon Scale
1
AutoGEM Control Display
Display AutoGEM Controls
Show Simulation Entities


Y
X
Z
WCS CSYS_DEF
End_Constraint: Dx:0 Dy:0 Dz:0
End_Constraint: Rx:0 Ry:0 Rz:0
Beam1
Concentrated_Load: Fx:0 Fy:-1000 Fz:
Concentrated_Load: Mx:0 My:0 Mz:0

III. ANALYSIS

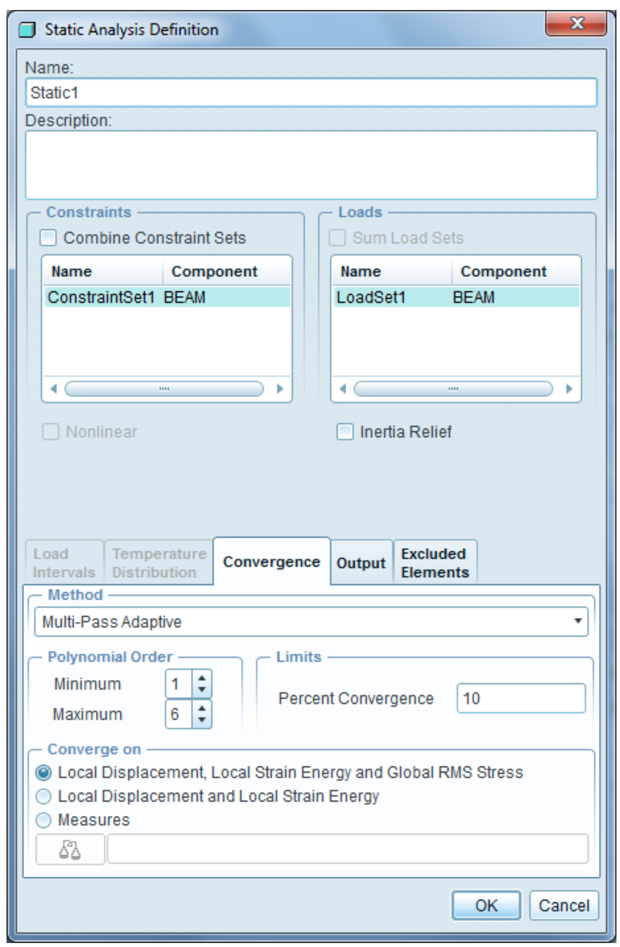
STEP 11: Analysis type definition. From the main menu, select **Analysis>Mechanica Analyses/Studies.** A dialog box will pop up prompting you to select the type of analysis. Select File>New Static" from the pull-down menu.



A Static Analysis Definition window will pop up. Enter the following:

Name: Static1
Description: Static analysis of rectangular beam (or whatever you want to describe your analysis)
Use Multi-Pass Adaptive option for convergence with 10%
Select Local Displacement, Local Strain Energy and Global RMS Stress as the convergence criteria

Click **OK**.



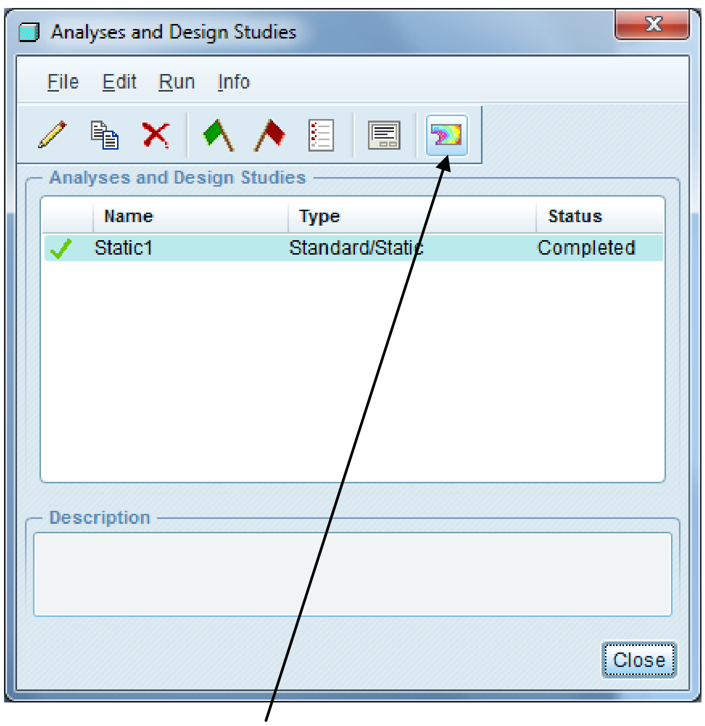
STEP 12: Carrying out the analysis. Select Start button in the Analyses and Design Studies dialog box. This will initiate the 'number crunching' process in Pro/M. Choose **NO** when you asked if you want to run interactive diagnostics. After starting the analysis, you may click on the Display Study Status button to "watch" the computing process in a pop-up window.

The analysis summary is given in the Summary window. Make sure to stroll through the result and understand the meaning of those values.

What is the value of von Mises stress from your run and what does it mean?

What is the value maximum deflection from your run?

You should do a quick hand calculation to check the result such as the maximum bending stress at the fixed end. How is this number (max bending stress) compared with the von Misses stress obtained from Pro/M?



When the calculations are done select the review result button in the Analysis and Design Studies dialog box.

Enter a name for the window. Click OK and ensure that the folder named "Static1" is selected (this is a folder in your working directory containing all the results from a particular analysis).

STEP 13: Shear and Moment Diagrams

In the "Result Window Definition" dialog, change Display type from Fringe to **Graph.**

In the **Quantity** menu, pull down and select **Shear & Moment** under Graph Ordinate (Vertical) Axis. Uncheck all except for Vy and Mz.

Make sure "**Beams**" is selected in the Graph Location group at the bottom.

Click on the **Select** button in the Graph Location group. A new window showing the beam model will pop up. Click on the beam. Make sure that you note the orientation of the beam according to the coordinate system shown in the window (in three colors). You may find it a bit clear when you rotate the model around. Click on the middle button on the mouse to exit this window. If prompted for the graph start, ensure that the constrained end of the beam is selected. If not, change this by pressing **Toggle.**

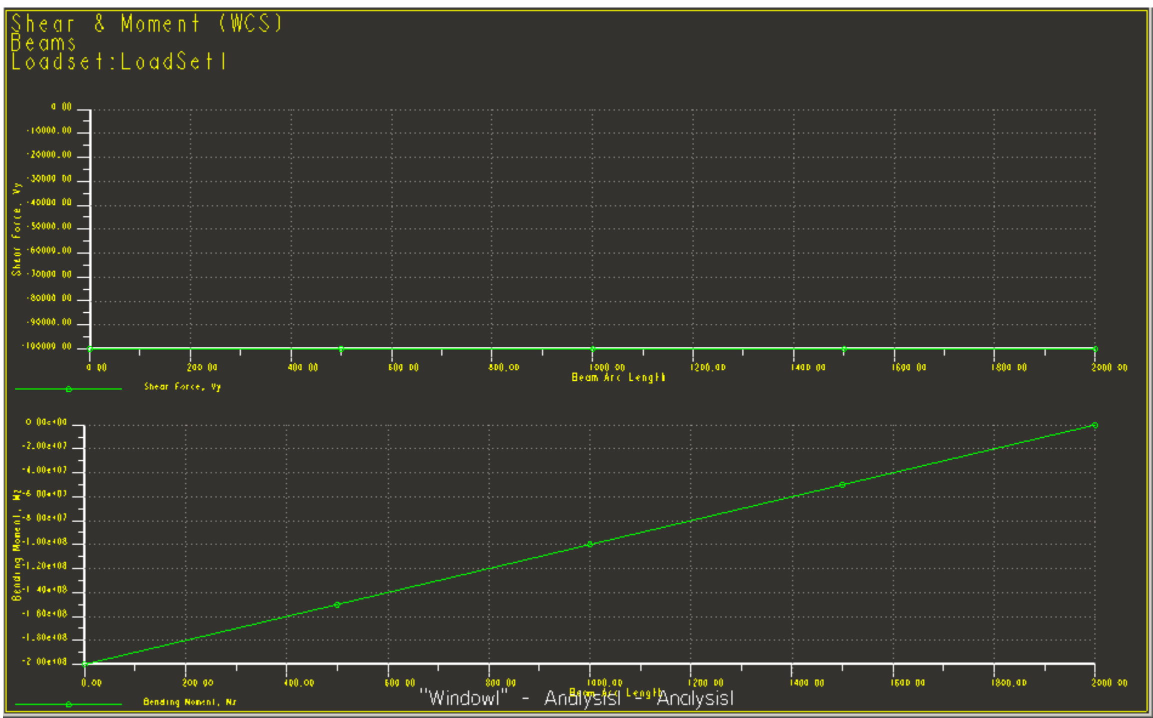
Leave Display in Graph mode and other default parameters unchecked.

Click on Accept button once complete.

An information dialog box will pop up. Click OK after reading the message.

Click **OK** then **Accept and Show** button.

The results should like this:

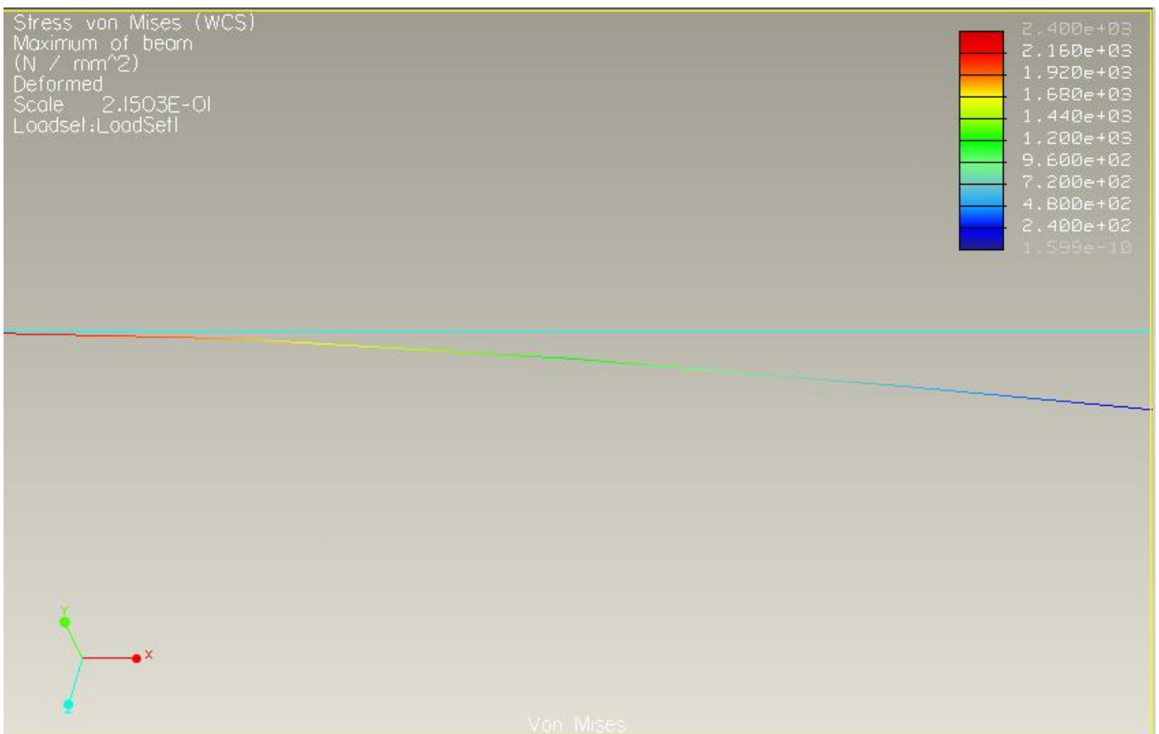


Make sure to check the values you see on these diagrams.

STEP 14: Viewing von Mises Stress.

To obtain stress information, follow the same steps above only here we select **Stress** as quantity under **Result Window Definition** dialog panel (accept the default Display type of **Fringe**). Also select **von Mises** the component pull-down menu. In the Display Location tab, select **ALL**.. Make sure **Deformed** box is checked in the Display Options tab and enter 10% for scale. To animate the result (from no stress to maximum stress), check the **Animate** box. Then click OK and Show.

You may rotate the model around to have a better view of the beam deflection. A screenshot result of von Mises plot is shown below.



You are encouraged to explore the result mode and experiment with different features and options such as displaying the maximum displacement and measurement of inertia etc.

To exit result display mode, select **Exit result** from **File** menu.

Additional Exercise: Use the same beam model you've created and apply a simply-support constraint at the mid point of the beam (constrain in the y-direction with respect to the WCS). Change the load from 100,000 N to 200,000 N.

Include in your report:

1- Plots of shear and moment diagrams
2- Plot of von Mises stress contour
3- Tabulated values of max. deflection and von Mises stress from FEM and your hand calculation (make sure to include *all* steps of your calculation in the appendix).
4- Comments on the results including *your* justification of errors, if any.
5- Results from the additional exercise specified below with all components listed in items 1 thru 4.