ME345: Modeling and Simulation Viewing Cross-Sectional Results of a Cantilever Beam with ANSYS[1]

Stevens Institute of Technology

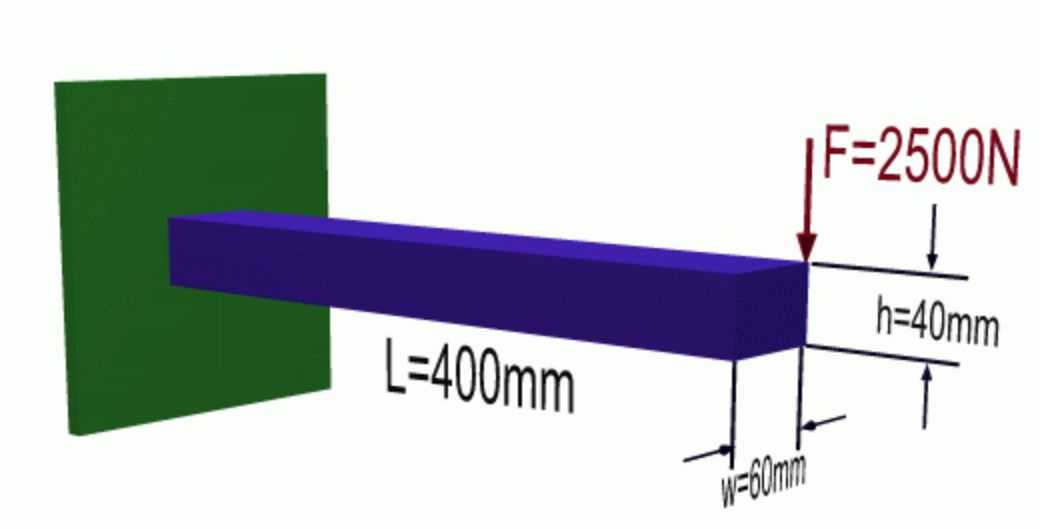
Professor Frank Fisher

Written by Ryan Oldja

Last Updated: 11/22/2015

Introduction

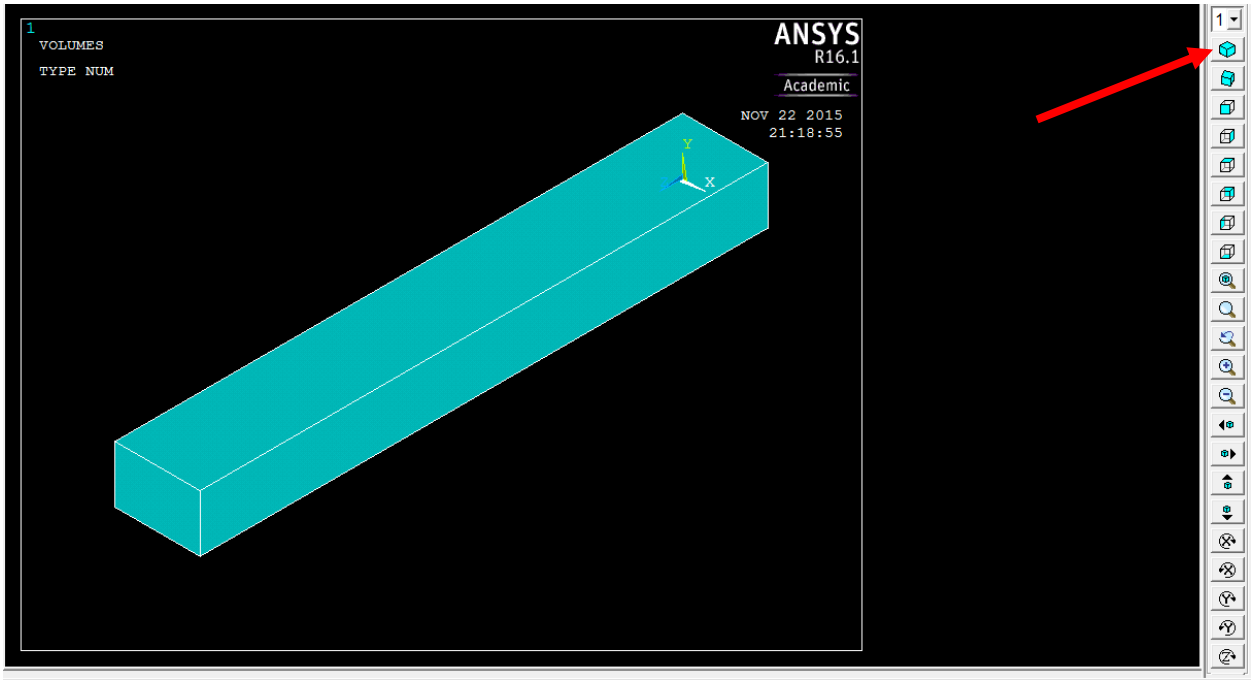
This tutorial was created using ANSYS 16.1. The purpose of this tutorial is to outline the steps required to view cross sectional results (deformation, stress, etc.) for the following example:



[1] This tutorial is based on a set of tutorials developed by Dr. Walied Moussa at the University of Alberta. See http://www.mece.ualberta.ca/tutorials/ansys/index.html

Pre-processing: Defining the Problem

1) Give the project a title
   a) Utility Menu > File > Change Title > "Cross-Sectional Results of a Simple Cantilever Beam"
2) Create solid beam
   a) Preprocessor > Modeling > Create > Volumes > Block > By 2 Corners & Z
      i) WP X = WP Y = 0
      ii) Width = 60mm
      iii) Height = 40mm
      iv) Depth = 400mm



3) Define the element type
   a) Preprocessor > Element Type > Add/Edit/Delete > Add… > Solid -- 20node 186 element
4) Define element material properties
   a) Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic
   b) In the window that appears, enter the following geometric properties for steel:

This tutorial is based on a set of tutorials developed by Dr. Walied Moussa at the University of Alberta. See http://www.mece.ualberta.ca/tutorials/ansys/index.html

i) Young's modulus EX = 200000

ii) Poisson's Ratio PRXY = 0.3

5) Define mesh size

a) Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size > Element edge length = 20

6) Mesh the volume

a) Preprocessor > Meshing > Mesh > Volumes > Free > 'Pick All'

Solution: Assigning Loads and Solving

7) Define analysis type

a) Solution > Analysis Type > New Analysis > Static

8) Apply constraints

a) Hint: Use Numbering to make selection easier

i) Utility Menu > PlotCtrls > Numbering > Area Numbers > Check 'On'

ii) Utility Menu > Plot > Areas

b) Then, Solution > Define Loads > Apply > Structural > Displacement > On Areas

i) We want to select the back face, on which the coordinate system lies. The desired face should be shown as 'A1'. This face can be selected by simply typing '1' in the input field of the open window and clicking OK

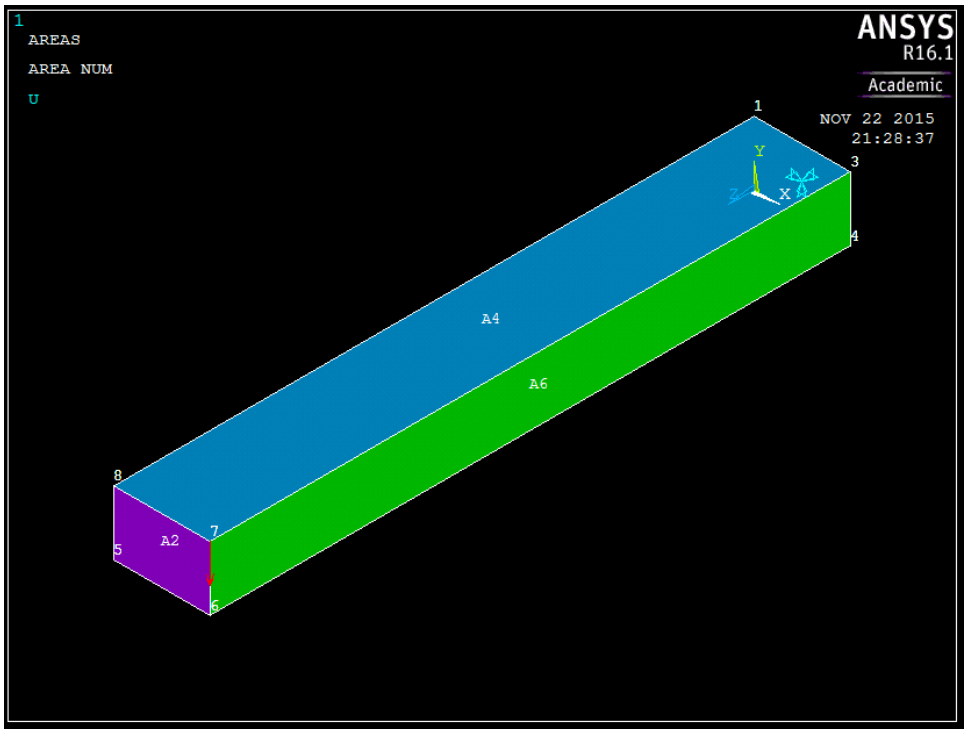
ii) Constrain ALL DOF to 0

9) Apply loads

a) Hint: Display model keypoints (similar to Area numbering)

b) Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

i) Apply a load of -2500N along FY on the front right-hand keypoint (keypoint #7)



10) Solve the system

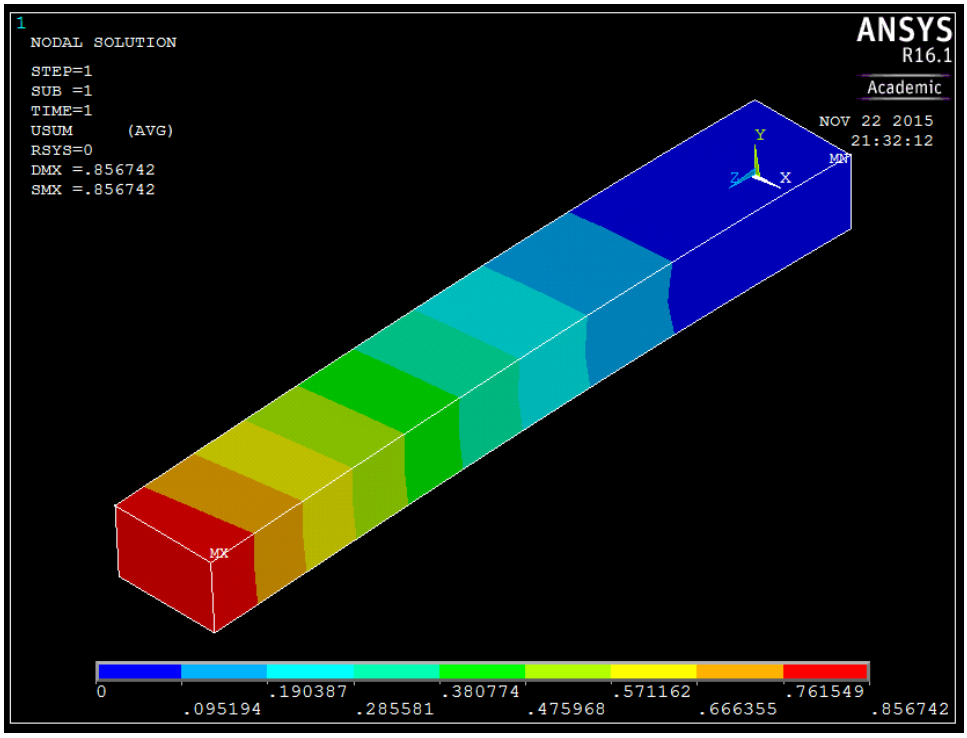
a) Solution > Solve > Current LS

Post-processing: Viewing the Results

11) Deflection – Entire Model

a) Before we begin selecting cross sections, let's view deflection of the entire model

b) General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution > Displacement Vector Sum



From here, one may wish to view several cross sections through the YZ plane.

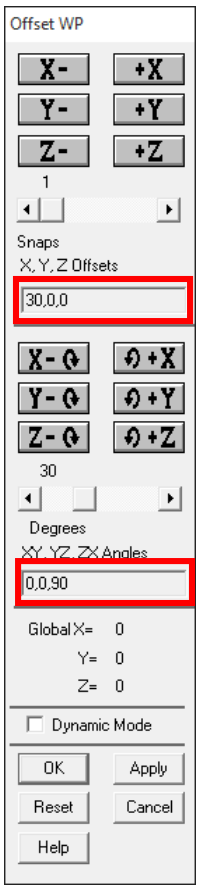
To illustrate how to take a cross section, let's take one halfway through the beam in the YZ plane…

12) Deflection – YZ Plane

a) First, offset the working plane to the desired position, halfway through the beam

   i) Utility Menu > WorkPlane > Offset WP by Increments

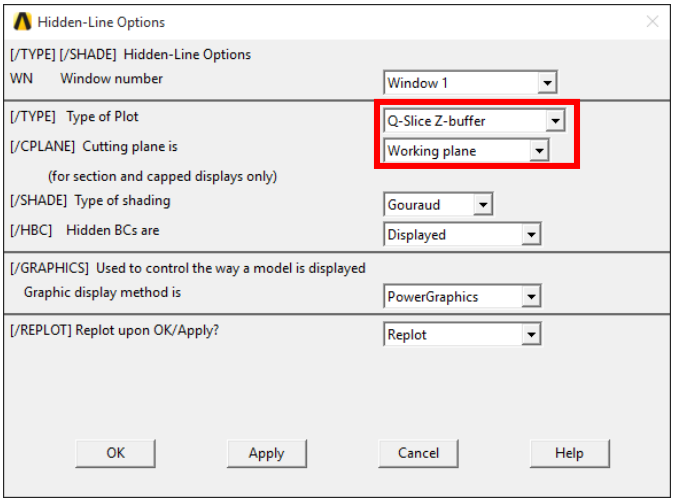
   ii) In the window that appears, increase Global X to +30 (Width/2) and rotate Y by +90 degrees. This can be done using the radio buttons, or by simply typing the desired offset in the input fields and clicking OK:



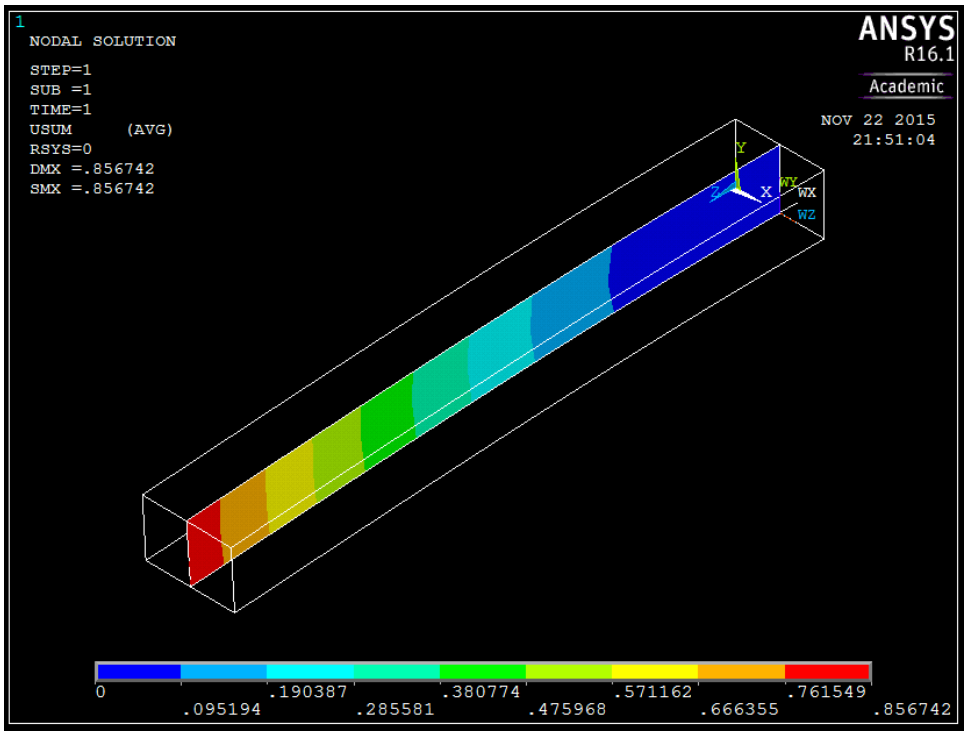
b) Select the type of plot and align the cutting plane with the working plane (note that in GUI, these two steps are combined)

i) Utility Menu > PlotCtrls > Style > Hidden-Line Options

ii) Fill in the window that appears as shown below to select 'Q-Slice Z-buffer' as 'Type of Plot' and 'Working Plane' as the 'Cutting Plane'



c) As desired, you should now have the following:



Note (DO NOT EXECUTE THESE COMMANDS): This procedure can also be repeated for any slice by typing the following command lines in the command prompt field:

```
WPOFFS,30,0, 0          ! Offset the working plane for cross-section view
WPROTA,0,0,90           ! Rotate the working plane
/CPLANE,1               ! Cutting plane defined to use the WP
/TYPE,1,8
PLNSOL,U,SUM,0,1
```

Also note that to realign the working plane with the active coordinate system, simply use:

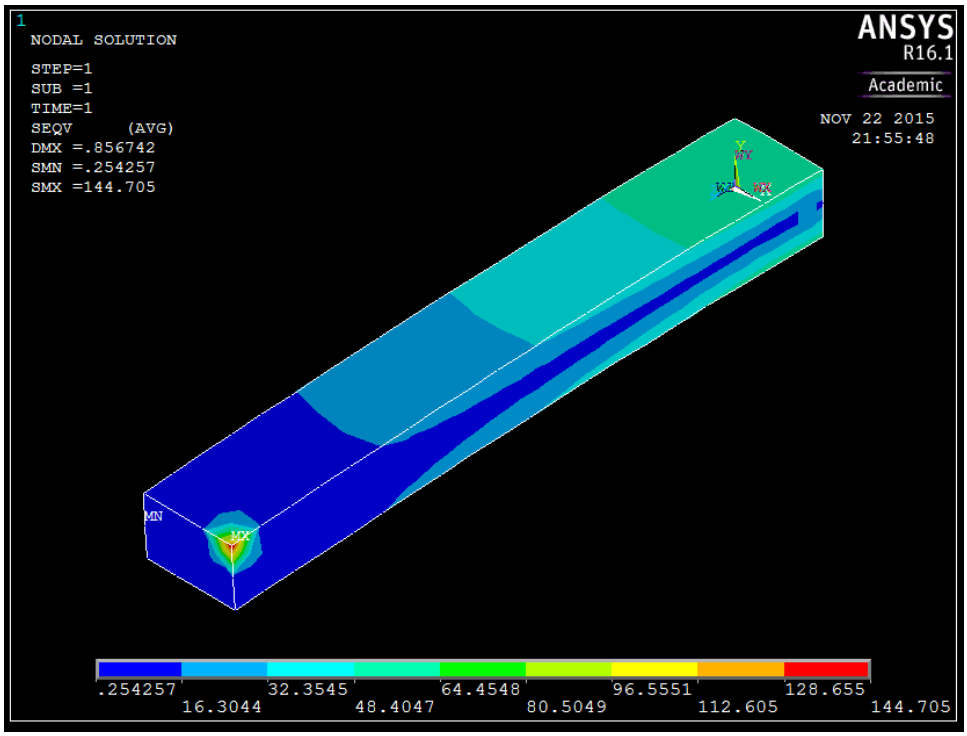
```
WPCSYS,-1,0
```

13) Equivalent Stress – Entire Model

a) First we need to realign the working plane with the active coordinate system.
   i) Utility Menu > WorkPlane > Align WP with > Active Coord Sys
   ii) NOTE: To check the position of the WP, select Utility Menu > WorkPlane > Show WP Status

b) Next we need to change plot style to the default setting (no hidden or section operations)
   i) Utility Menu > PlotCtrls > Style > Hidden Line Options... And change the 'Type of Plot' to 'Non-hidden'

c) General Postproc > Plot Results > Contour Plot > Nodal Solu > Stress > von Mises Stress



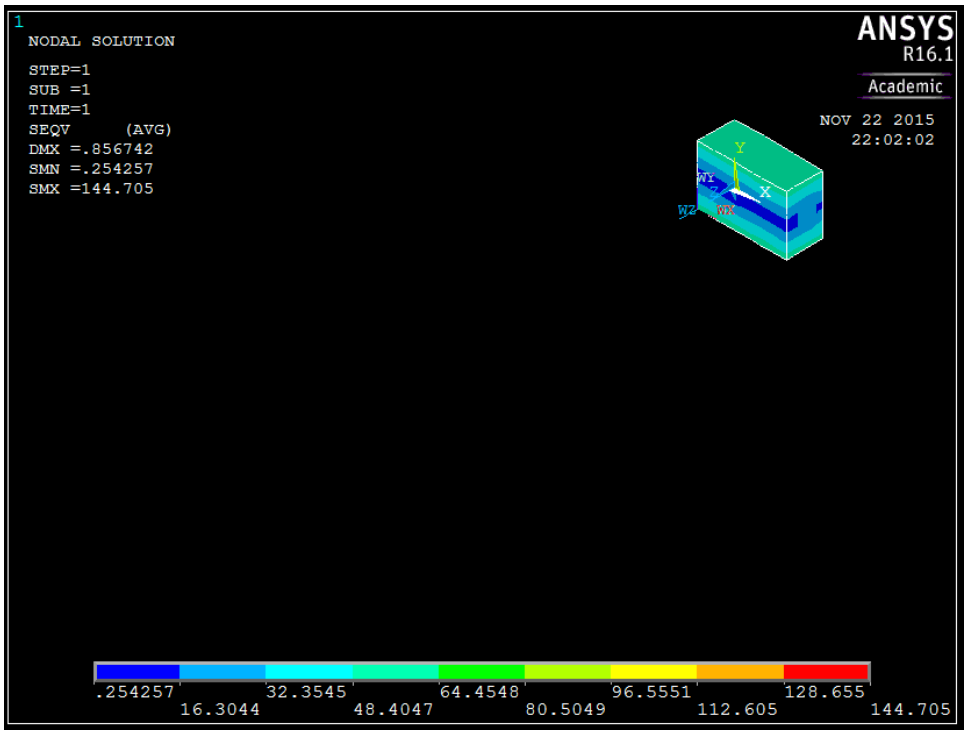
14) Equivalent Stress – Capped Cross-Section

a) First, offset the working plane to the desired position, halfway through the beam
   i) Utility Menu > WorkPlane > Offset WP by Increments
   ii) In the window that appears, increase Global Z to +25 (Length/16)

b) Select the type of plot and align the cutting plane with the working plane (note that in GUI, these two steps are combined)
   i) Utility Menu > PlotCtrls > Style > Hidden-Line Options

ii) Change ‘Type of Plot’ to ‘Capped Hidden’

c) You should now see the following:



15) Results Animation

a) Now, for something a little more impressive, let's show an animation of the Von Mises stress through the beam

i) First, revert back to the ‘non-hidden’ display style

(1) Utility Menu > PlotCtrls > Style > Hidden Line Options…

(2) Change ‘Type of Plot’ to ‘Non-hidden’

ii) Now, we can define the results animation

(1) Utility Menu > PlotCtrls > Animate > Q-Slice Contours

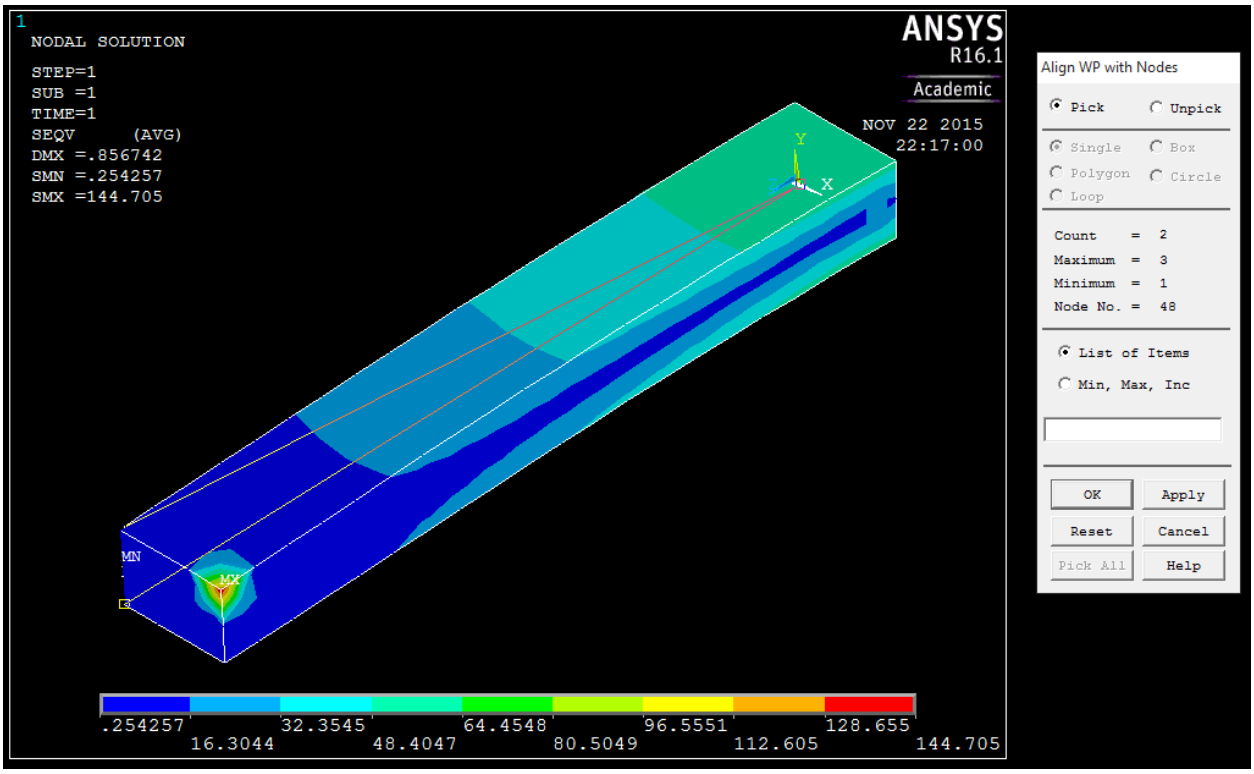
(2) In the window that appears, just change the Item to be contoured to 'Stress' 'von Mises'

iii) You will then be asked to select 3 nodes; the origin, the sweep direction, and the Y axis

(1) First, select the bottom left-hand corner at the back of the model, where the coordinate system lies

(2) Next, select the bottom left-hand corner at the front of the model to define the sweep direction of the animation (along the Z-direction)

(3) Finally, select the top left-hand corner at the front of the model to define the Y-axis



iv) You should now see an animated version of the stress contour slices through the beam, scanning front to back along the Z-direction.

b) For more information on how to modify the animation, type help *ancut* into the command line.