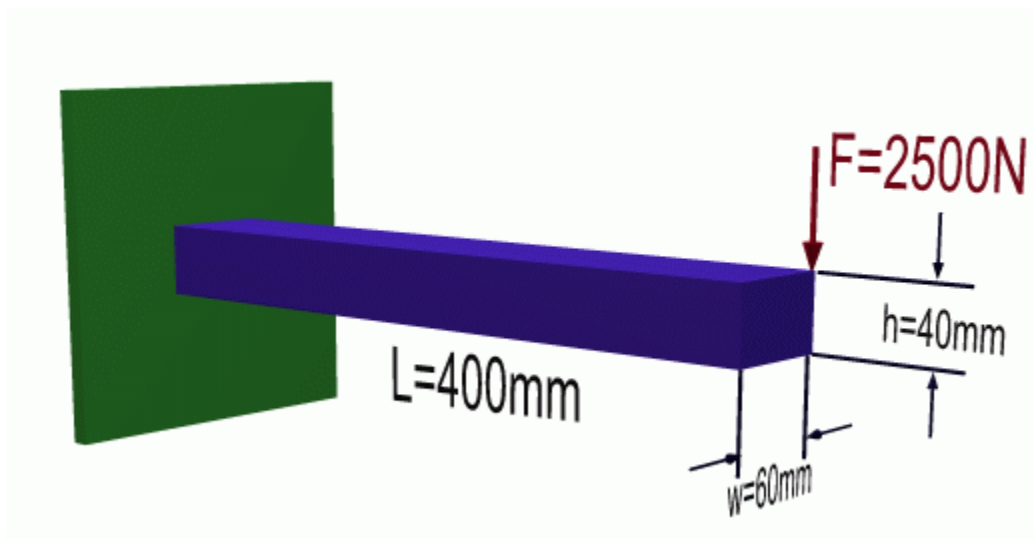


# Viewing X-Sectional Results

## Introduction

This tutorial was created using ANSYS 6.1 The purpose of this tutorial is to outline the steps required to view cross sectional results (Deformation, Stress, etc.) of the following example.



## Preprocessing: Defining the Problem

### 1. Give example a Title

Utility Menu > File > Change Title ...

/title, Cross-Sectional Results of a Simple Cantilever Beam

### 2. Open preprocessor menu

ANSYS Main Menu > Preprocessor

/PREP7

### 3. Create Block

Preprocessor > Modeling > Create > Volumes > Block > By 2 Corners & Z

BLC4, 0, 0, Width, Height, Length

Where: Width: 40mm  
Height: 60mm  
Length: 400mm

### 4. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the SOLID45 (3D Structural Solid) element. This element has 8 nodes

each with 3 degrees of freedom (translation along the X, Y and Z directions).

#### 5. Define Element Material Properties

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for steel:

- i. Young's modulus EX: 200000
- ii. Poisson's Ratio PRXY: 0.3

#### 6. Define Mesh Size

Preprocessor > Meshing > Size Cntrl > ManualSize > Global > Size  
esize, 20

For this example we will use an element size of 20mm.

#### 7. Mesh the volume

Preprocessor > Meshing > Mesh > Volumes > click 'Pick All' vmesh, all

---

## Solution: Assigning Loads and Solving

#### 1. Define Analysis Type

Solution > New Analysis > Static  
ANTYPE, 0

#### 2. Apply Constraints

Solution > Define Loads > Apply > Structural > Displacement > On Areas  
Fix the left hand side (should be labeled Area 1).

#### 3. Apply Loads

Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints  
Apply a load of 2500N downward on the back right hand keypoint (Keypoint #7).

#### 4. Solve the System



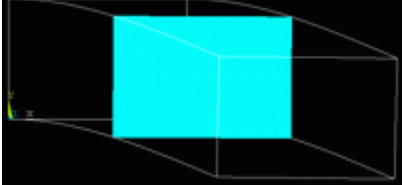
Solution > (-Solve-) Current LS  
SOLVE

---

## Postprocessing: Viewing the Results

Now since the purpose of this tutorial is to observe results within different cross-sections of the colume, we will first outline the steps required to view a slice.

- Offset the working plane for a cross section view (**WPOFFS**)
- Select the TYPE of display for the section (**/TYPE**). For this example we are trying to display a section, therefore, options 1, 5, or 8 are relevant and are summarized in the table below.

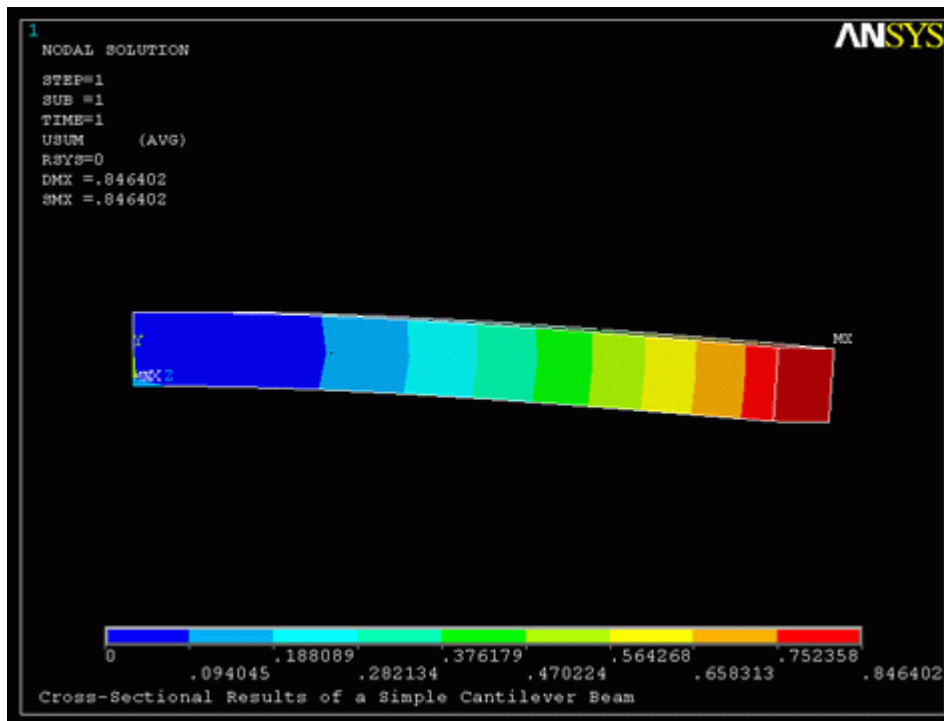
Type	Description	Visual Representation
SECT or (1)	Section display. Only the selected section is shown without any remaining faces or edges shown	
CAP or (5)	Capped hidden display. This is as though you have cut off a portion of the model and the remaining model can be seen	
ZQSL or (8)	QSLICE Z-buffered display. This is the same as SECT but the outline of the entire model is shown.	

- Align the cutting plane with the working plane (`/CPLANE`)

## 1. Deflection

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

- Select: **General Postproc > Plot Results > Contour Plot > Nodal Solu**

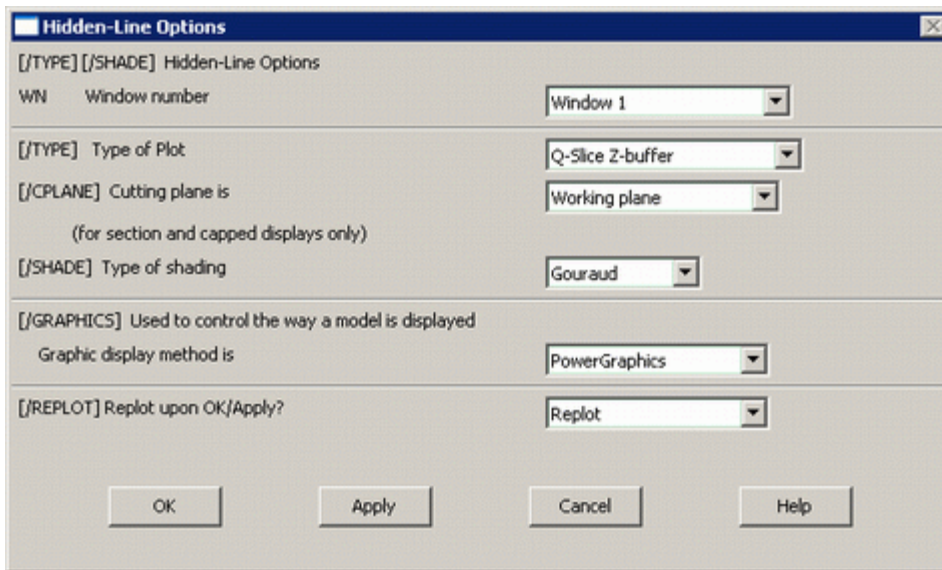


From this one may wish to view several cross sections through the YZ plane.

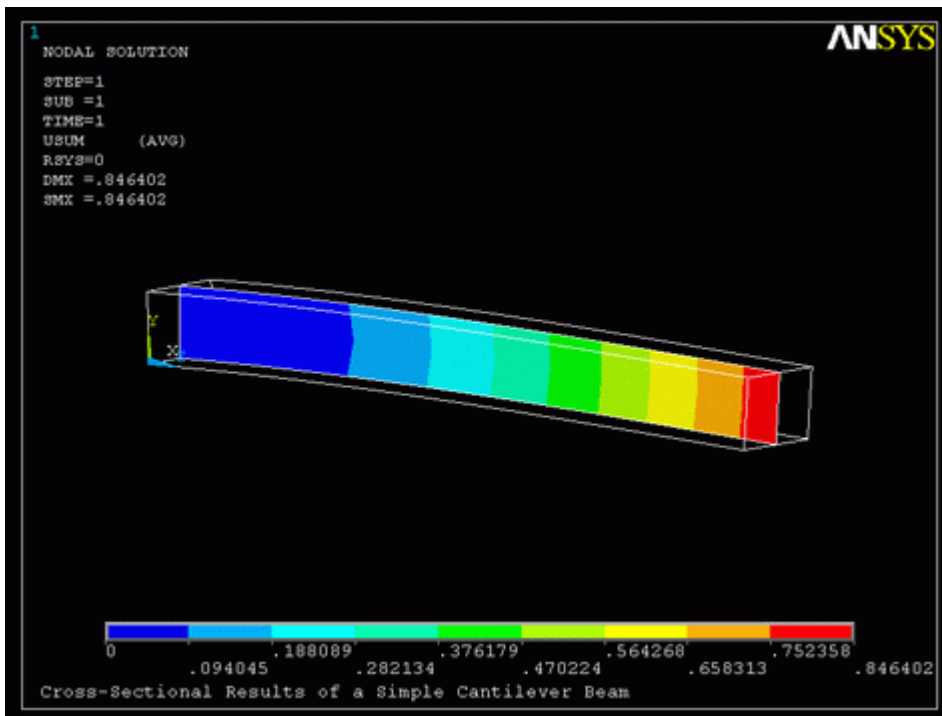
To illustrate how to take a cross section, let's take one halfway through the beam in the YZ plane

- First, offset the working plane to the desired position, halfway through the beam  
 Select: **Utility Menu > WorkPlane > Offset WP by Increments**  
 In the window that appears, increase Global X to 30 (Width/2) and rotate Y by +90 degrees
- Select the type of plot and align the cutting plane with the working plane (Note that in GUI, these two steps are combined)  
 Select: **Utility Menu > PlotCtrls > Style > Hidden-Line Options**

Fill in the window that appears as shown below to select **/TYPE=ZQSL** and **/CPLANE=Working Plane**



As desired, you should now have the following:



This can be repeated for any slice, however, note that the command lines required to do the same are as follows:

```

WPOFFS,Width/2,0,0      ! 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**

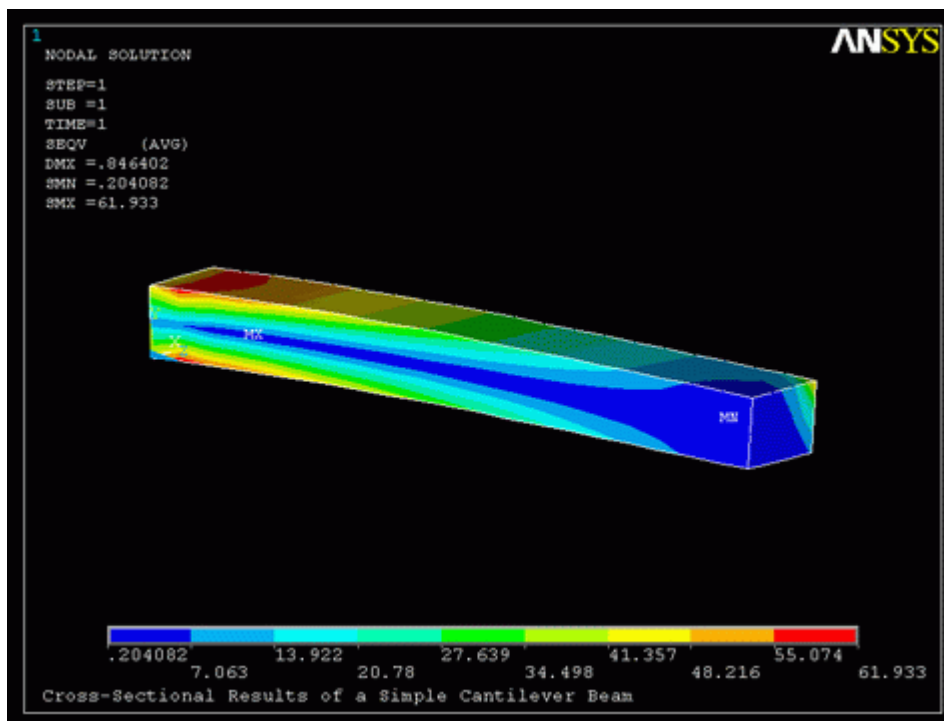
## 2. Equivalent Stress

Again, let's view stresses within the entire model.

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

Next we need to change **/TYPE** to the default setting(no hidden or section operations). Select: **Utility Menu > PlotCtrls > Style > Hidden Line Options...** And change the 'Type of Plot' to 'Non-hidden'

- o Select: **General Postproc > Plot Results > Contour Plot > Nodal Solu > Stress > von Mises**

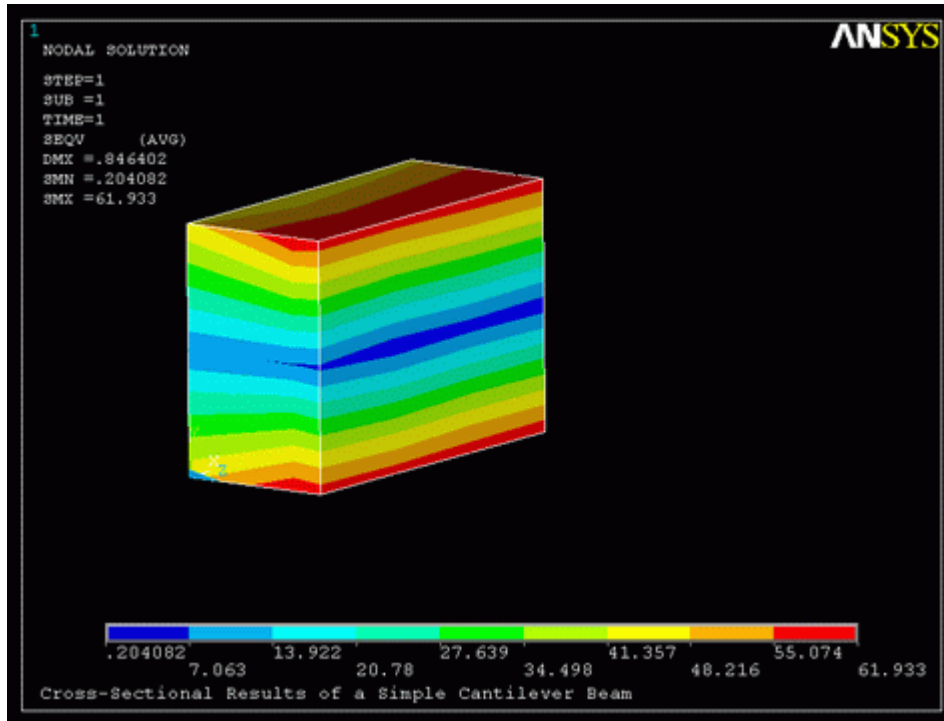


Let's say that we want to take a closer look at the base of the beam through the XY plane. Because it is much easier, we are going to use command line:

```
WPOFFS,0,0,1/16*Length    ! Offset the working plane
/CPLANE,1                 ! Cutting plane defined to use the WP
/TYPE,1,5                 ! Use the capped hidden display
PLNSOL,S,EQV,0,1
```

Note that we did not need to rotate the WP because we want to look at the XY plane which is the default). Also note that we are using the capped hidden display this time.

You should now see the following:



### 3. Animation

Now, for something a little more impressive, let's show an animation of the Von Mises stress through the beam. Unfortunately, the ANSYS commands are not as user friendly as they could be... but please bear with me.

- Select: **Utility Menu > PlotCtrls > Animate > Q-Slice Contours**
- In the window that appears, just change the Item to be contoured to 'Stress' 'von Mises'
- You will then be asked to select 3 nodes; the origin, the sweep direction, and the Y axis. In the graphics window, select the node at the origin of the coordinate system as the origin of the sweep (the sweep will start there). Next, the sweep direction is in the Z direction, so select any node in the z direction (parallel to the first node). Finally, select the node in the back, bottom left hand side corner as the Y axis.

You should now see an animated version of the contour slices through the beam. For more information on how to modify the animation, type `help ancut` into the command line.