

Changing Graphical Properties

Introduction

This tutorial was created using ANSYS 7.0 This tutorial covers some of the methods that can be employed to change how the output to the screen looks. For instance, changing the background colour, numbering the nodes, etc.

Since the purpose of this tutorial is not to build or analysis a model, please copy the following code and paste it into the input line below the utility menu.

```
finish
/clear
/title, Changing Graphical Properties
/prep7
```

```
K,1,0,0
K,2,100,0
L,1,2
```

```
et,1,beam3
r,1,100,833.333,10
mp,ex,1,200000
mp,prxy,1,0.3
```

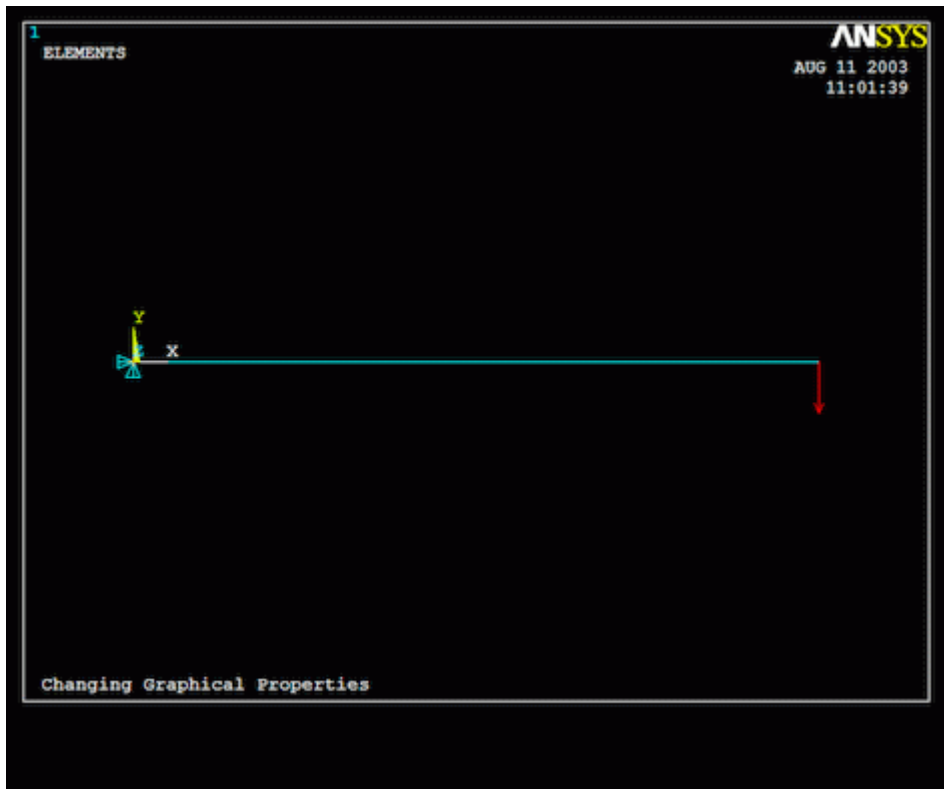
```
esize,5
lmesh,all
```

```
finish
/solu
```

```
antype,0
dk,1,all,all
fk,2,fy,-100
```

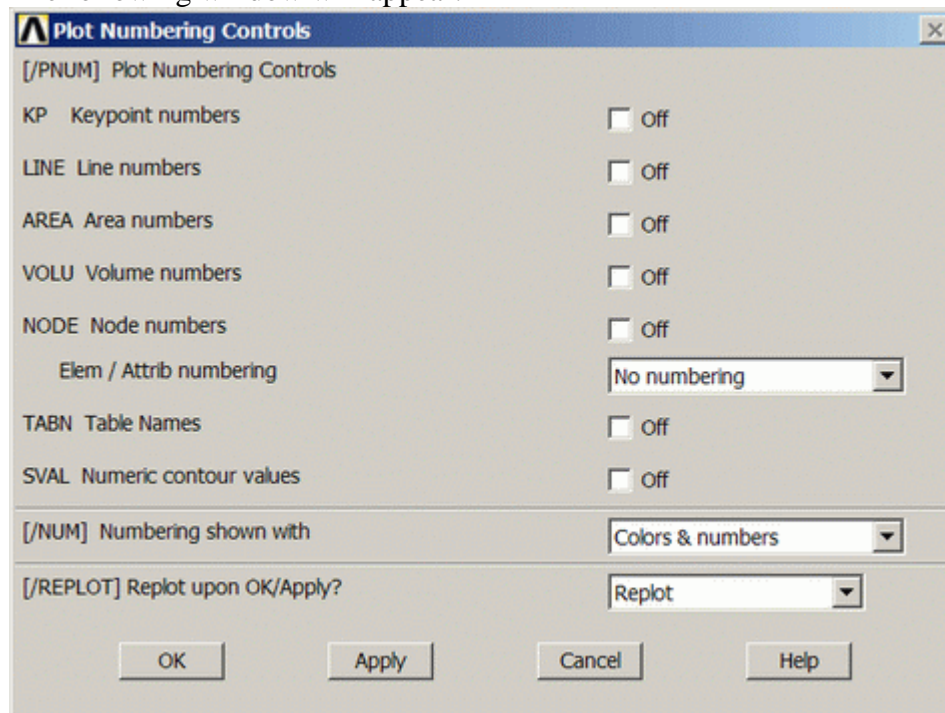
```
solve
finish
```

You should obtain the following screen:



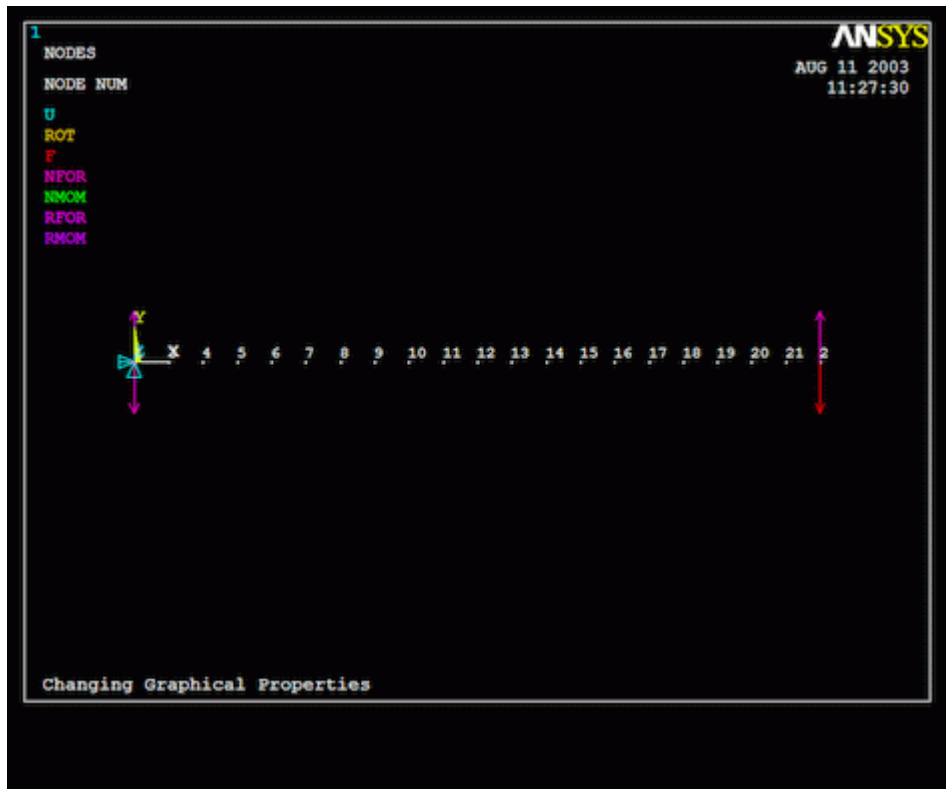
Graphical Options

1. **Number the Nodes**
Utility Menu > PlotCtrls > Numbering...
The following window will appear:



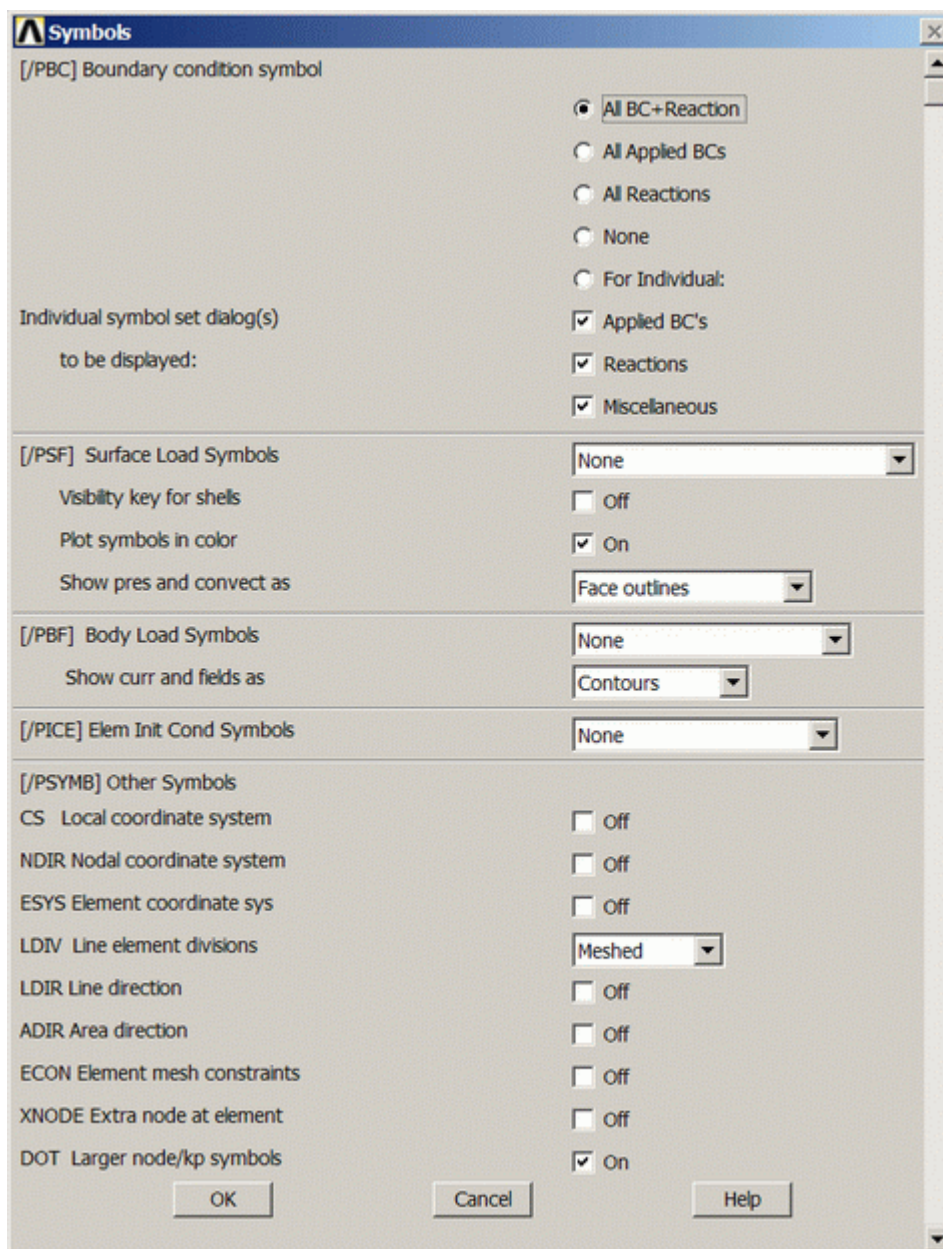
From this window you can select which items you wish to number. When you click OK, the window will disappear and your model should be numbered appropriately. However, sometimes the numbers won't show up. This could be because you had previously selected a plot of a different item. To remedy this problem, select the same item you just numbered from the **Utility > Plot** menu and the numbering will show up.

For instance, select the node numbering and plot the nodes. You should get the following:



As shown, the nodes have been numbered. You can also see some other information that ANSYS is providing. The arrows on the left and the right are the force that was applied and the resulting external reactive forces and moments. The triangles on the left are the constraints and the coordinate triad is also visible. These extra symbols may not be necessary, so the next section will show how to turn these symbols off.

2. **Symbol Toggles**
Utility Menu > PlotCtrls > Symbols

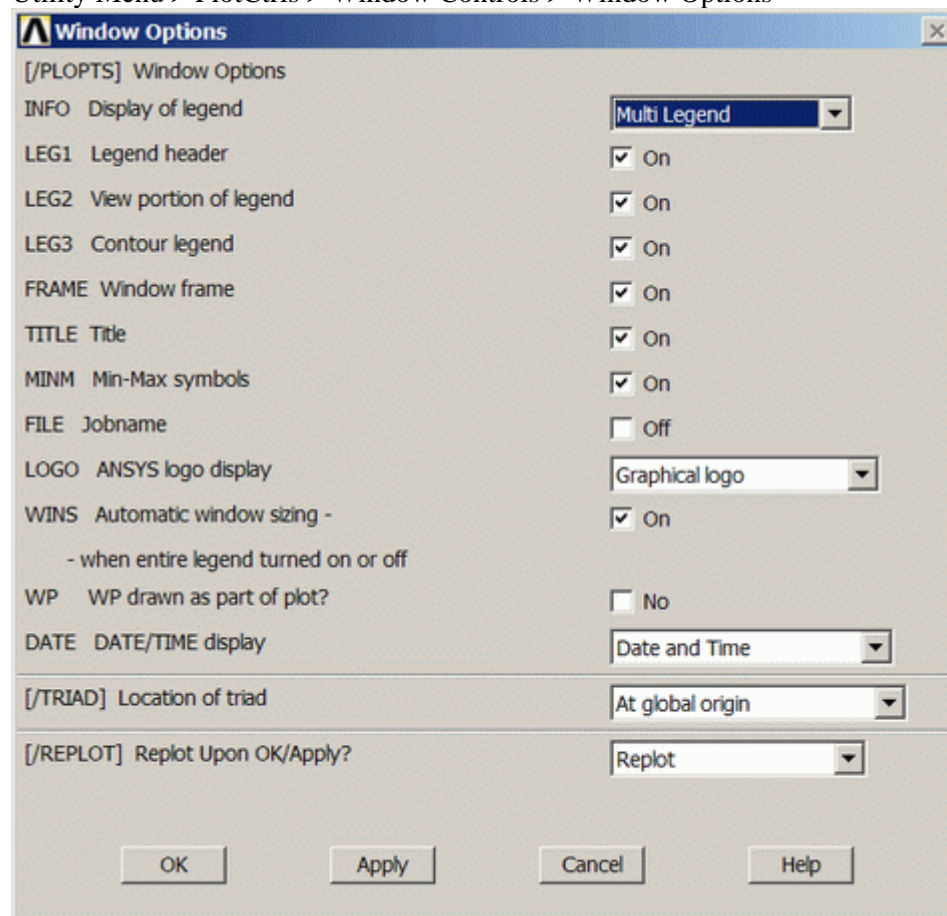


This window allows the user to toggle many symbols on or off. In our case, there are no Surface or Body Loads, or Initial Conditions, so those sections won't be used. Under the **Boundary conditions** section, click on **None** to turn off all the force and reaction symbols. The result should be as follows:



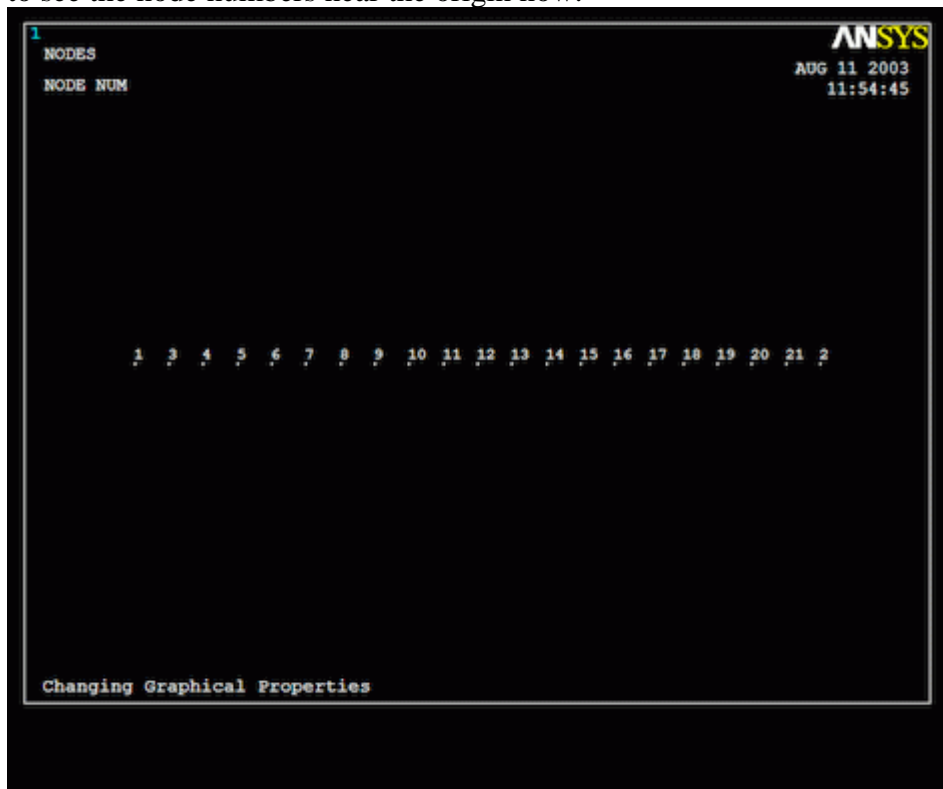
3. Triad Toggle

Utility Menu > PlotCtrls > Window Controls > Window Options



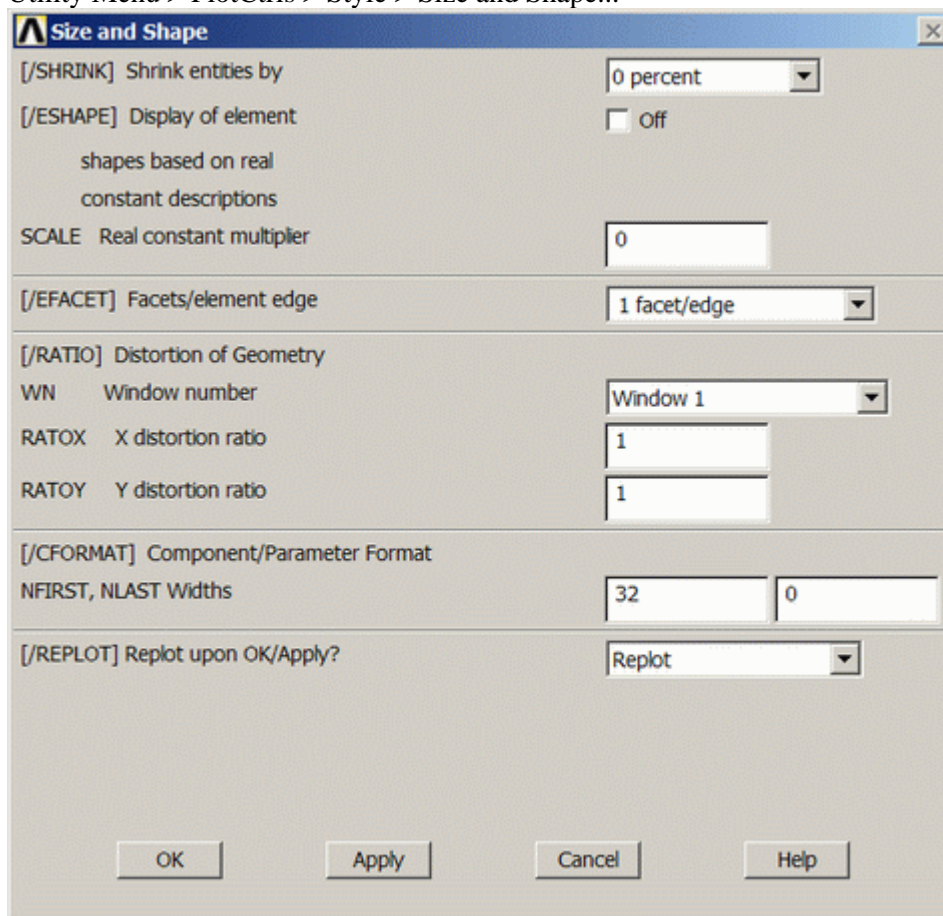
This window also allows the user to toggle many things on and off. In this case, it is things associated with the window background. As shown in the window, the legend or title can be turned off, etc. To turn off the triad, select **Not Shown** from the Location of triad

drop down menu. The following output should be the result. Notice how it is much easier to see the node numbers near the origin now.



4. Element Shape

Utility Menu > PlotCtrls > Style > Size and Shape...



When using line elements, such as BEAM3, it is sometime difficult to visualize what the elements really look like. To aid in this process, ANSYS can display the elements shapes based on the real constant description. Click on the toggle box beside [/ESHAPE] to turn on element shapes and click OK to close the window.

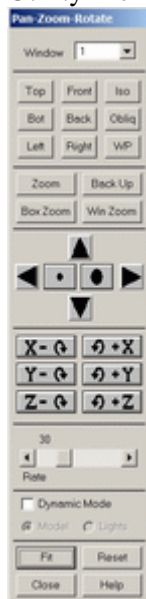
If there is no change in output, don't be alarmed. Recall we selected a plot of just the nodes, thus elements are not going to show up. Select **Utility Menu > Plot > Elements**. The following should appear.



As shown, the elements are no longer just a line, but they have volume according to the real constants. To get a better 3-D view of the model, you can change the view orientation.

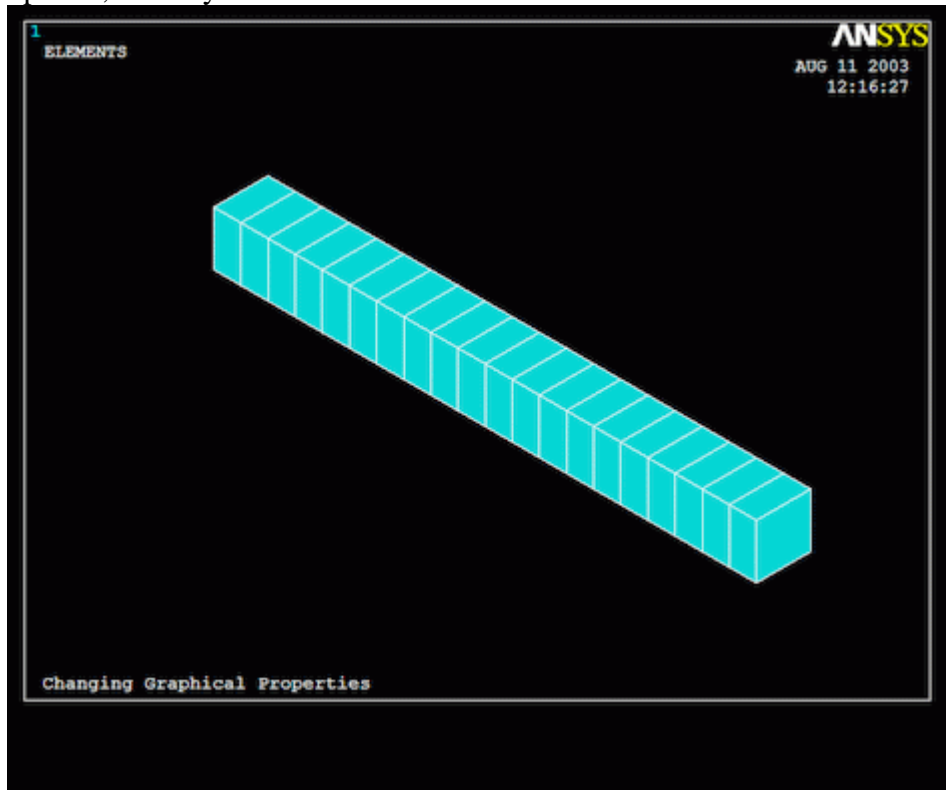
5. View Orientation

Utility Menu > PlotCtrls > Pan Zoom Rotate...



This window allows the user to rotate the view, translate the view and zoom. You can also select predefined views, such as isometric or oblique. Basic rotating, translating and zooming can also be done using the mouse. This is very handy when you just want to

quickly change the orientation of the model. By holding the **Control** button on the keyboard and holding the **Left mouse button** the model will translate. By holding the **Control** button on the keyboard and holding the **Middle mouse button** the model will zoom or rotate on the plane of the screen. By holding the **Control** button on the keyboard and holding the **Right mouse button** the model will rotate about all axis. Using these options, it's easy to see the elements in 3-D.



6. Changing Contours

First, plot the deformation contour for the beam.

General Postproc > Plot Results > Contour Plot > Nodal Solution > DOF Solution > USUM

If the contour divisions are not appropriate, they can be changed.

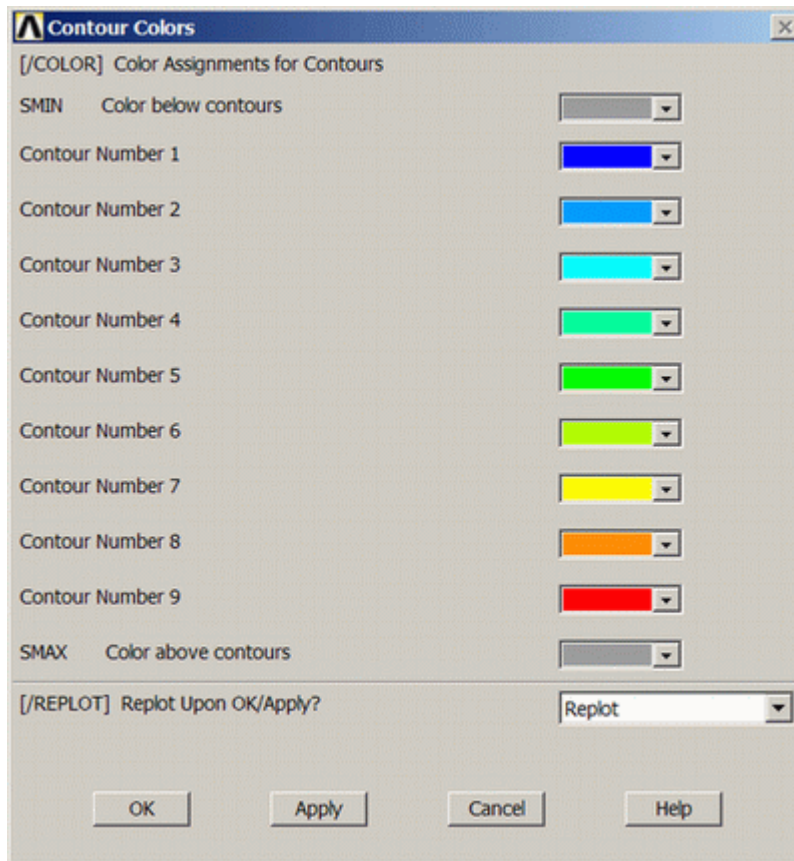
Utility Menu > PlotCtrls > Style > Contours

Either **Uniform** or **Non-uniform Contours** can be selected. Under uniform contours, be sure to click on **User specified** if you are inputting your own contour divisions. Under non-uniform contours, you can create a logarithmic contour division or some similar contour where uniform divisions don't capture the information you desire.

If you don't like the colours of the contour, those can also be changed.

Utility Menu > PlotCtrls > Style > Colours > Contour Colours...

The colours for each division can be selected from the drop down menus.

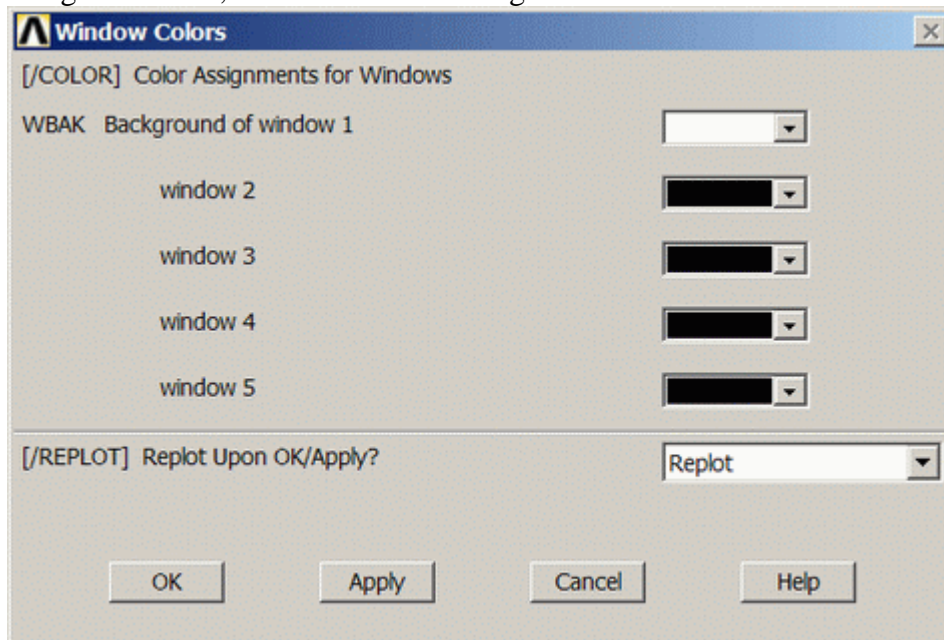


7. Changing Background Colour

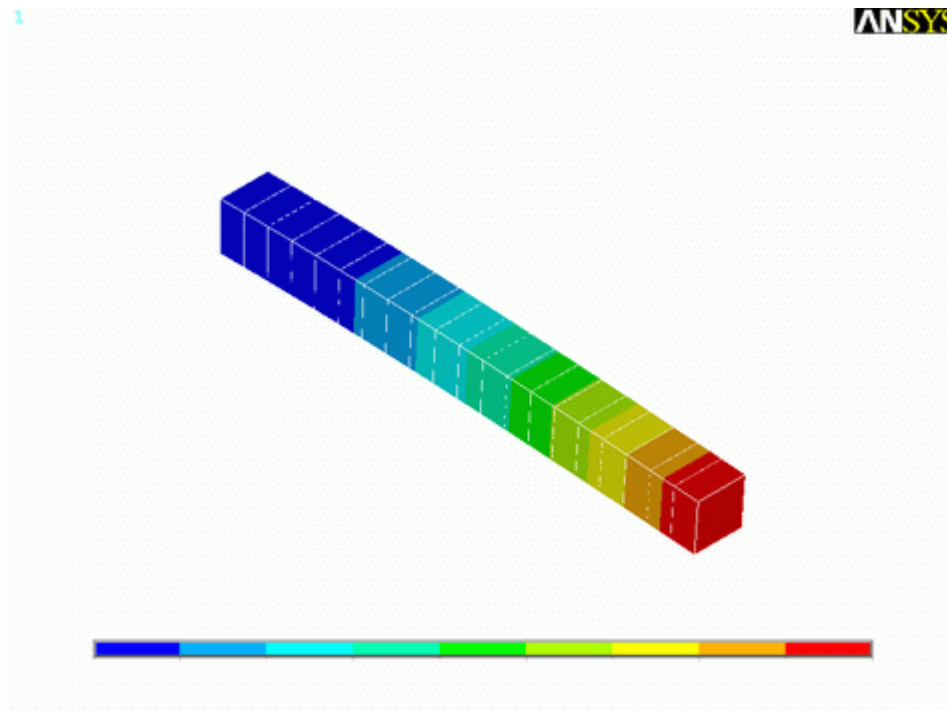
Perhaps you desire to use a plot for a presentation, but don't want a black background.

Utility Menu > PlotCtrls > Style > Colours > Window Colours...

Select the background colour you desire for the window you desire. Here we are only using Window 1, and we'll set the background colour to white.



The resulting display is shown below. Notice how all the text disappeared. This is because the text colour is also white. If there is information that needs to be added, such as contour values, this can be done in other graphic editors. To save the display, select **Utility Menu > PlotCtrls > Capture Image**. Under the File heading, select Save As...



There are lots of other options that can be used to change the presentation of data in ANSYS, these are just a few. If you are looking for a specific option, the PlotCtrls menu is a good place to start, as is the help file.