**ANSYS TUTORIAL**

Start ANSYS from the Start Menu. (All Programs 🡪 CAD and Engineering Software 🡪 ANSYS 14.5 🡪 Mechanical APDL 14.5 (ANSYS) ) You will be guided through the menus of ANSYS to perform various tasks. The procedures below will help you set up your first case for under a point load. Most commands will originate from the ANSYS Main Menu located on the left part of the screen. For each dimensional quantity you input, do not type the dimensions. So long as you are consistent, ANSYS will assume you are working in SI units.

Preprocessor 🡪 Element Type 🡪 Add/Edit/Delete

 Click the “Add…” button to add the type of structure we want to model. Select

 “Beam” from the left menu, and “3 node 189” from the right menu. This tells

 ANSYS that we want to use beam elements, each with 3 nodes for a 3-D, finite

 strain beam. The nodes will represent the beginning, end, and midpoint of each

 element. Ensure that the “Element type reference number” is set to 1. Click

 “OK” and close the “Element Types” window.

Preprocessor 🡪 Material Props 🡪 Material Models

 We need to provide material properties of our beam. When the window opens, on

 the right-hand side, select Structural 🡪 Linear 🡪 Elastic 🡪 Isotropic. We are

 assuming that our beam behaves in a linearly-elastic manner in all directions. In

 the window that automatically opens, we will set Young’s modulus (EX) to

 1.0e11 Pa and the Poisson ratio (PRXY) to 0.3. Click “OK” and close the

 materials models window.

Preprocessor 🡪 Sections 🡪 Beam 🡪 Common Sections

 We need to select a cross-sectional shape for our beam. From the “Sub-Type”

 drop-down menu, select the I-beam. In the “Name” field, type “Beam”. You now

 need to give the cross-section its dimensions. Our beam will be symmetric with

 all widths equal to 0.1 m and all thicknesses equal to 0.001 m. Enter these values

 into the 6 fields below the picture of the I-beam. Click “Apply” then “OK”.

Now we must create our model.

Preprocessor 🡪 Modeling 🡪 Create 🡪 Keypoints 🡪 In Active CS

 Your beam will be 2 m long, with the origin being the fixed end. Type “1” for the

 Keypoint number and (0,0,0) as its coordinate. Click “Apply”. Repeat, typing

 “2” for the keypoint number, and (2,0,0) for its coordinate. Click “Apply” then

 “OK”. Two points should appear on the graphic window. (Point 1 may be

 obscured by the coordinate axes.)

Preprocessor 🡪 Modeling 🡪 Create 🡪 Lines 🡪 Lines 🡪 Straight Line

 Now connect the two points. When the window pops up, use the cursor to select

 Point 1 followed by Point 2. Your line should turn blue. Click “OK”.

Preprocessor 🡪 Meshing 🡪 Mesh Tool

 This will mesh your beam. In the pop-up window, click “Set” on the “Lines”

 row. With the cursor, select the line in the graphics window and click “OK” in

 the window that pops up. In the window that opens, enter “30” in the “No. of

 element divisions” field. This will create 30 elements out of your beam. Click

 “OK”. Your line should now be dashed. Now click “Mesh” towards the bottom

 of the “Mesh Tool”. Select the dashed line with the cursor and click “OK” in the

 window that pops up. To view the full beam type “/eshape,1” followed by

 “/replot” in the command line at the top of the screen. Click the isometric view

 button on the right side of the screen for a better view. Close the mesh tool.

Solution 🡪 Analysis Type 🡪 New Analysis

 Here you are selecting the governing equations for the type of model you want

 to solve. We will solve a “Static” problem. Select it and click “OK”.

Solution 🡪 Define Loads 🡪 Apply 🡪 Structural 🡪 Displacement 🡪 On Keypoints

 Now we apply boundary conditions and loads. Select Point 1 with the cursor

 and click “OK”. This will be a fixed point (no deflection or rotation). Hightlight

 “All DOF” (degrees of freedom) and set the VALUE to 0. Check the KEXPND

 box to yes. Click “OK”.

Solution 🡪 Define Loads 🡪 Apply 🡪 Structural 🡪 Force/Moment 🡪 On Keypoints

 Select Point 2. Click “OK”. Change the direction of the force to FZ in the drop

 down menu. Give it a constant VALUE of 500 N. Click “OK”.

Solution 🡪 Solve 🡪 Current LS

 Now we can solve problem. Click “OK” in the box that appears and then “Close”

 once the solution is done. You can also close the “/STATUS Command”

 window.

General PostProc 🡪 Plot Results 🡪 Contour Plot 🡪 Nodal Solu

 To view your solution, click “DOF Solution” and “Z-Component of

 displacement” followed by Apply. The graphics window now shows a contour

 plot of the beam deflection. Units are in meters. Repeat this process, but this

 time click “Stress” and “X-Component of stress”. Repeat again for the

 “von Mises” stress. Make either a color or grey scale plot for each of these to

 turn in for your assignment. Use the printer button at the top of the screen.

 You can select other properties to view as well.

Repeat the tutorial, experimenting with different settings, geometries, etc.