Structural Analysis in ANSYS

Exercise problem description
A long, slender beam with rectangular section of 10mm x 10mm and length of 100mm has one fixed end and one free end. A point force, 10N, is applied on the free end.

Method
Static analysis in ANSYS with Solid element types.

Build your component in SolidWorks
1. Draw the desired beam in the SolidWorks
2. Export the file into IGES format
   File → Save as → Change the file type to “IGES (*.igs)” →Click “Options” → Change the “Surface representation/System preference" from "Standard" to "ANSYS" → Click "ok" to close "Export Options" → Choose the folder and the name the file is saved → Click "ok" (Choose "All bodies" if the dialog box "Export" poped up)

Import file in ANSYS window
1. File Menu → Import → IGES → Click ok for the poped up dialog box → Click "Browse" and choose the file saved from SolidWorks → Click ok to import the file

ANSYS procedure
1. Define the analysis as a structural analysis
   Main Menu → Preference → check “Structural” → ok
2. Select the element type as Solid 45
   Main Menu → Preprocessor → Element Type → Add/Edit/Delete →
   Click “Add” in the element dialog box →
   Choose “Solid” under “Structural” in the left scroll box of element types→
   Choose “Brick 8node 45” in the right scroll box →
   Click “ok” to close the library →
   Click “close” to close the element dialog box
3. Input the material properties
   Main Menu → Preprocessor → Material Props → Material Models →
   In the “Define Material Model Behavior” dialog box, double click “Structural” on the right box → double click “Linear” → double click “Elastic” → double click “Isotropic” → Input “1.9e11” in the box for EX (Elastic Modulus) → Input “0.29” in the box for PRXY (Possion’s Ratio) → Click “ok” to add the properties → double click “Density” → Input “8000” in the box for DENS → Click “ok” to add the property → Close the “Define Material Model Behavior” dialog box
4. Mesh the structure
   Main Menu → Preprocessor → Meshing → Mesh Tool →
   In the “Mesh Tool” dialog box, check “Smart Size” and move the size down to fine “1” → Click on "Mesh" → In the dialog box of pick, click "Pick All" → After meshing, click on "Refine" in the "Mesh Tool" dialog box → Click "Pick All" → Change the level of refinement to "2" → Click ok → Click "Close" to close the Mesh Tool dialog box

5. Apply the load
   Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Areas → Pick the fixed end area by mouse click (You can go to File Menu → PlotCtrls → Pan Zoom Rotate, to find the area, check "Dynamic Mode" to rotate the model by mouse) → Click "ok" → Choose "All DOF" and put "0" as the value → Click "ok"
   Main Menu → Solution → Define Loads → Apply → Structural → Force/Moment → On Keypoints → Pick two vertex points on the edge the force applies → Click "ok" → Choose "FY" as the direction and Input -5 into the value → Click "ok"

6. Solve the problem
   Main Menu → Solution → Current LS → Click "ok" → Click "yes" if the warining message poped up → Click "close" after the solution is done and close the window of commands

7. Review the results
   Main Menu → General Postproc → Plot Results →Contour Plot → Nodal Solu → Choose the results you want to review, for example, DOF Solution → Displacement vector sum → Click "ok"
8. Save the file
   File Menu → Save as Jobname.db

Compare the simulation results with Solidworks
Repeat the whole process with another element type, solid → Tet 10node 187
File Menu → Clear & Start New → Click "ok" → Click "Yes" → Repeat Import file and ANSYS procedure for the new element.