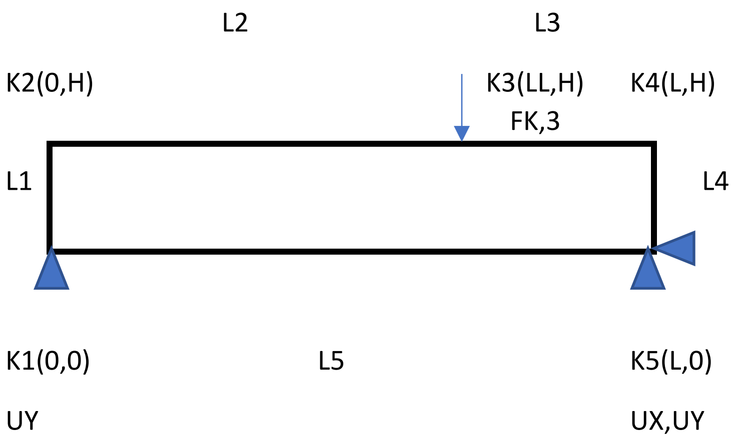
ANSYS Trouble-shooting tips:

1) Avoid the temptation to sit at the computer and create the model without first doing the paper work!
2) Avoid the temptation to sit at the computer and create the model without first doing the paper work!
3) Avoid the temptation to sit at the computer and create the model without first doing the paper work!
4) See rules 1 through 3.

Paperwork:

1) Create a sketch of the object
2) On the sketch, identify geometric features such as coordinates of key points
3) On the sketch, show the applied loads
4) On the sketch, number the key points and lines connecting them
5) On the sketch, identify how you wish to constrain it



(The notes on the model may or may not be the precise script format, but must communicate unambiguously)

6) Identify in writing the element type you are using, and identify the nodal DOF's (from ANSYS Element description; see the help library)
7) Identify in writing what units you will be using. If you are using N and mm, then a stress of 70 would be 70MPa. If you are using N and m, stress of 70 would be 70Pa.
8) Identify in writing, the required material properties using the appropriate units.
9) Create the appropriate script and double check it.

ANSYS – after **completing <u>all</u> of the above**, load APDL and cut-paste your script FOR THE GEOMETRY only – do not include meshing, yet. View the model to make sure the geometry looks correct. Then cut-past the meshing and boundary conditions. Keep an eye on the screen while the model is being analyzed – look for any warning messages that may appear and then disappear. Then cut and paste the entire script. Look for any warning messages that may appear and then disappear

Here are some common problems and their causes:

- You see the "Solution is done!" icon (ya! This is good) --- but darn it, when you want to plot the stress contours it says "SPLOT not available" (or something like that). This is likely due to you running the student edition and/or using your laptop. Don't worry, hopefully the results are there, but in a different place. Select RESULTS VIEWER from the side-menu, select Element Solution>select desired stress to view. This should work. If it doesn't, you may need to use a University computer in the student computer lab.
- You see the "Solution is done!" icon and you can plot the stresses, but they are all zero! This is likely due to the loads not actually being applied. You may have attempted to apply them at a location without a node. oops.
- You do **NOT** get the message "Solution is done". The most common problem causing this is insufficient constraints. Modify your script – where ever you have applied a specific constraint (such as UX) change it to constrain ALL. Try running the model again. If it works, then you know the original problem was lack of constraints. Go back and modify your constraints making sure you constrain the model based on the DOF for nodes. Line elements (BEAM188, BEAM189) have 6 DOF nodes – you must prevent the model from translating in all 3 directions and rotating in all 3 axes. This does not mean you need to apply 6 DOF constraint to any single node – it means the entire model must be preventing from moving in all 6 DOF. The example shown above (simply supported beam) shows that by preventing translation in X and Y at one node and in Y only at the other node prevents X, Y translation <u>and</u> rotation (ROTZ) of the beam. Be sure your final model has the appropriate constraints (to match the real-world structure you are trying to analyze).
- Most other problems are caused by some subtle typographical error. Make sure you have the appropriate spelling of commands, appropriate number of commas, etc., etc., etc.