ANSYS - Truss Step 9

Step 9: Validate the results

Do not assume that if you are able to obtain a solution from ANSYS, it is bound to be correct. It is ***very important*** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Simple Checks

- Does the deformed shape look reasonable and agree with the applied boundary conditions? We checked this in step 8.
- Do the reactions at the supports balance the applied forces for static equilibrium? We checked this also in step 8.

Refine Mesh

The results obtained from FEA analysis depend on the mesh. An important step in the analysis is to make sure that the mesh resolution is adequate for the desired level of accuracy. This is done by refining the mesh and comparing results obtained with different levels of mesh resolution.

In our truss example, however, a truss member has to be modeled as a ***single*** *LINK1* element. If we use multiple *LINK1* elements to model a single truss member, these elements can rotate freely with respect to each other since they are essentially linked through pin joints. This violates physical reality and is one of the few cases where we'll avoid refining the mesh since it leads to an incorrect result.

Compare with Theory

Results should be compared with appropriate theoretical results whenever possible. In most cases, one would use theory to obtain order-of-magnitude estimates rather than to make a head-to-head comparison since presumably FEA is being used because a theoretical solution is not available. In this case, however, one can easily determine the forces in the truss members using the method of joints from statics. I'd recommend that you take a few minutes during commercials on your favorite TV show to calculate the forces and compare them with your ANSYS results.

Exit ANSYS

Utility Menu > File > Exit

Select ***Save Everything*** and click ***OK***.

This is just a quick introduction to ANSYS to give you a flavor of what a full-fledged engineering package looks like. If it felt unfriendly or cumbersome, you are not alone; I went through this myself (otherwise, congratulations! you are a genius). It takes some getting used to. Believe it or not, it gets a lot easier to use with time. You have a lot of years ahead of you to gain the experience necessary to harness the power of finite-element analysis. All the ANSYS features including the underlying theory are documented online and can be accessed using **Utility Menu > Help**. There are tutorials available in the documentation which are also useful.

Go to Problem Set 1

See and rate the complete Learning Module

Go to all ANSYS Learning Modules