Using Finite Element codes

Examples in structural analysis with ANSYS

The following information on each component of a structure to be studied is needed:
1) geometry
2) material properties
3) boundary conditions
4) loading conditions

When the model is built (Preprocessor), it is possible to start the calculation of the results (Solution) and when the results are obtained, they can be displayed in different ways (Postprocessor).

Geometry (Preprocessor)

a) Preliminary
   Depending on the required level of refinement of the results (first pass design or detailed analysis), and on the type of loads to be applied (point forces, pressure…) first determine the type of elements that you are going to use (1D, 2D, 3D, solid elements, shell elements, axisymmetric elements…). Generally, the geometry needs to be constructed before the mesh is defined and nodes are created. This allows the user to check whether or not refining the mesh improves the results (mesh sensitivity analysis). Sometimes, nodes and then elements can be created directly. Regardless, the model should always be made as small as possible to save computer resources: all the symmetries (geometry and loading) should be used. Keep meshing in mind when building the geometry so as to avoid singularities. Use your experience in CAD to build the geometry.
b) Keypoints
c) Lines
   Created by keypoints
d) Surfaces
   Created by keypoints or lines (by contour or by sweeping line around axis)
e) Volumes
   Created by keypoints, lines or surfaces, or as preformed volumes.
f) Boolean operations (additions, subtractions, fillets…) may be needed.

Meshing (Preprocessor)

a) Preliminary
   A common mistake is to want to create elements one by one as you would create the geometry. This is not how it works, except perhaps for beam elements.
b) Line/surface/volume seeding
c) Choice of element type
d) Specification of which material properties will be used with those elements
e) Meshing

Material properties (Preprocessor)

a) Preliminary
   Material properties may be assigned at the element level or higher (part, whole model). If anisotropy is an issue, it is essential to properly define the directions of anisotropy in the model. It may be necessary to define local coordinate systems, or make sure that the sides of the elements are aligned with the preferred directions.
b) Isotropic linear elastic
   Young's modulus, Poisson's ratio
c) Orthotropic linear elastic
   Elastic moduli, Poisson's ratios, shear moduli

Boundary conditions (Preprocessor)

How is the structure held in place? What are the prescribed displacements or rotations? Are there symmetry/antisymmetry conditions to be enforced? It may be necessary to define local coordinate systems. At the element level, what are the degrees of freedom involved in the boundary conditions?

Loading (Preprocessor)

Is it a static, dynamic or transient analysis? Determine the point loads, loads per unit length, surface loads (pressure), volume loads (gravity). See the documentation on the element used in the analysis.

Analysis (Solution)

a) Linear solution
   The solution is computed in only one run (no iterations).
b) Geometric and/or material nonlinearities
   Because of nonlinear stress-strain curves, contact or stress-stiffening phenomena, iterations are needed for convergence of the solution, because the stiffness of the structure at one given iteration depends on the displacements found at the previous iteration. The total load is first divided into load steps, and iterations (or substeps) are run until convergence criteria are satisfied. This can be done automatically or semi-automatically.

Displaying the results (Postprocessor)

The results of the analysis can be displayed in terms of displacements/rotations, stress and strains, and many options are available for the output (local or global coordinate systems). In case of a symmetric structure, even if, say, ¼ of the structure has been modeled, ANSYS can display the results for the whole structure.

Checking the results

There are different complementary ways of verifying that your results are correct. First, make sure that the program did what you had in mind (resolve any issue with software usage per se). Then double-check your input data. Should the results display symmetries? Check your results against predictions from closed-form equations for the same or simplified problem. Are the units correct? After this first pass of verification is complete, refine the mesh (e.g. divide the element edges by 2) and check by how much the results vary between the coarse and refined meshes. If too large a variation is noticed (typically more than a few percent), continue mesh refinement until satisfied. If the solution does not converge, examine the cause of the problem (element type, approximate theory, etc…). From a design point of view, will there be adverse practical consequences from an un-converged solution? If so, a clear warning must be sent.

Other considerations

a) Working directory
   ANSYS will be run from the working directory selected at the beginning of the session. It may be necessary to create that directory before starting, and ANSYS has to be restarted to use another directory.
b) File management
   ANSYS gives a name to all the files that it needs during the session. The default file name is "file", and the extension varies according to the use of the file. The default name for a database is "file.db". That file is overwritten everytime a new session is open. Therefore, it is essential to save your current database under a distinct name (File, Save As, "filename.db"). The database contains everything that has been done so far with the model.
c) Re-opening a database
   Resume from "filename.db".
d) Starting from a script file (ANSYS commands)
   Read input from "filename.txt". "Filename.txt" is a text file that is created outside of ANSYS, using text editors like Notepad or Wordpad for example. This is by far the most powerful method for parametric studies, and gives you total control on the study parameters.