MECH 151, Finite Element Analysis

Solving Finite Element Problems with MATLAB

(Mechanics problems, static)

Since we're using MATLAB, we will use these steps instead of those outlined in Chapter 1 of the text.

Create functions for each element type

You only do this once for each element time. Once these functions are finished and have been check out, you can re-use them without change.

1. A function to calculate element stiffness matrices k (global coordinates). Example: `truss2k`.
2. A function to recover internal forces and stresses after displacements have been calculated. Example: `truss2s`.

Create a script to solve the problem

Often you can re-use much of the code from a previous problem to solve a new problem.

1. Create arrays defining the mesh
   (a) By hand:
      - Enter node coordinates in the `nodes` array.
      - Enter element connections in the `elems` array¿
      - Plot the model if desired.

   (b) Using `mesh2d`:
      - Enter super-node coordinates in the `snodes` array.
      - Enter edge information in the `edges` array.
      - Call `mesh2d` to generate `nodes` and `elems`.
2. Define element properties (geometric and material).
3. Enter loads in the `loads` array and support conditions in the `bc` array.
4. Create a global stiffness matrix full of zeros. Assemble it by doing the following for each element
   (a) Generate an element stiffness matrix
   (b) Add the element stiffness matrix to the accumulated global stiffness matrix using DOF indexing.
5. Plot the element if desired.

6. Create a global load vector full of zeros. For each row in the `loads` array, use DOF indexing to add the load to the accumulated load vector
7. Solve the equilibrium equations
   (a) Create a global displacement vector full of zeros.
   (b) Create a list of free (unsupported) DOF called `freedof`. For each row of the `bc` array, remove the corresponding entry in `freedof`.
   (c) Solve the equilibrium equations for the free DOF
8. Tabulate global displacements if desired.
9. Compute and tabulate reaction forces if desired.
10. Generate and report element results if desired. For each element
    (a) Generate internal results such as stresses, forces, or moments, as appropriate for the element type.
    (b) Format and display results in the command window or a text file.
    (c) Plot the deformed shape, if desired, using a suitable scale to exaggerate displacements.