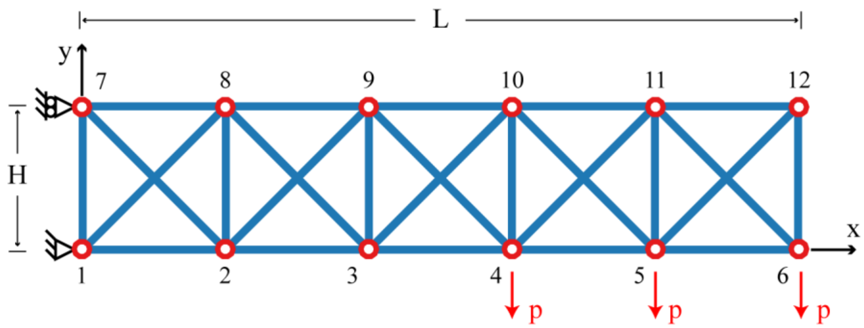
**Programming assignment #3**

Write a MATLAB function that solves for the displacement a 2D truss structure, such as the one pictured below, and computes the axial stress within each truss element.




Each of the truss elements is composed of hollow aluminum tubing with an outer diameter of 20 mm and a wall thickness of 4 mm. Assume that the density of the steel is $\rho$ = 8050 kg/m$^3$ and E = 210 GPa. Consider the weight of each element as point forces acting equally on the element nodes (½ of the weight of each element should be distributed to each of the element nodes). Additionally, concentrated forces act along the negative y-direction on the three right-most nodes along the bottom of the truss structure with a magnitude of p, specified in Newton.

**Instructions for programming and assignment submission:**

- Code for generating the mesh is supplied in a MATLAB file named "**make_truss.m**". The mesh connectivity and nodal coordinates are returned in a MATLAB structure (for more details, see http://www.mathworks.com/help/matlab/structures.html). Do not submit this function along with your assignment, i.e., treat **make_truss** as if it were a built-in MATLAB function.
- The file **must** define a function of the same name as the file name (but without the ".m") , e.g.

```
function [d, sig] = asurite_hw3(L, H, n, p)
    [mesh] = make_truss(L, H, n);
    % Define stiffness matrix.
    % Calculate external nodal forces (both point forces and from mass)
    % Apply displacement boundary conditions.
    % Solve system of equations.
    % Calculate stress of each element.
end
```

- The order of the input variables and output variables must not be changed.
- **The input variables are:**
  - **L, H:** the length and height of the truss, specified in meters.
  - **n:** the number of segments along the truss (in the image above, n = 5). Note: you can assume that n > 2.
  - **p:** magnitude of the three concentrated forces, specified in Newton.
- **The output variables are:**
  - **d:** 2N×1 array of nodal displacements in the format $[u_{1x}, u_{1y}, u_{2x}, u_{2y}, \ldots, u_{Nx}, u_{Ny}]^T$

- **sig:** $N_e \times 1$ array of element stresses in the format $[\sigma^{(1)}, \sigma^{(2)}, \sigma^{(3)}, \ldots, \sigma^{(Ne)}]^T$

**Helpful code for debugging:**

The following function will plot the deformed structure with elements colored by their axial stress value.

**<u>Please comment out the call to this function prior to submitting your code.</u>**

```
function [] = plot_solution(mesh, d, sig)
% PLOT_NUMBERED_MESH  plots truss mesh with numbered nodes.
%   PLOT_NUMBERED_MESH(mesh) plots mesh.
    scale_factor = 1;
    x = mesh.x + scale_factor*[d(1:2:end), d(2:2:end)]';
    figure(1); clf;
    hold on;
    colormap(jet());
    cmap = jet;
    v = (sig - min(sig)) / (max(sig) - min(sig));
    for e = 1:length(mesh.conn)
        c = mesh.conn(:,e);
        r = interp1(linspace(0,1,length(colormap)), cmap(:,1), v(e));
        g = interp1(linspace(0,1,length(colormap)), cmap(:,2), v(e));
        b = interp1(linspace(0,1,length(colormap)), cmap(:,3), v(e));
        line(x(1,c), x(2,c), 'linewidth', 5, 'color', [r,g,b]);
    end
    scatter(x(1,:), x(2,:), 50, 'filled', 'markeredgecolor', 'k', ...
            'markerfacecolor', 'w', 'linewidth', 2);
    axis equal;  axis off;
    colorbar;
    caxis([min(sig), max(sig)]);
end
```