Flowchart of "driver.m"

1. Giving the boundary condition

↓

2. Dived the mesh

(How many node, element, nodes of element & equations)

↓

3. Generate local node index with global index matrix & Generate equation respetively to points

↓

4. Set the weight & quadrature points

↓

5. Generate zero matrix with same size of Stiffness maritx And Force vector for each element

↓

6. Add each element respectively to form the Stiffness matrix and load vector