1. Generate Nodes for Vega FEM solver

   Run the code below :

   "FSI_Chengjun\LauderMembraneVibration\inputFiles_v5_membrane_iteration_forceCorrection**generateNodesForMembrane.c**"

   generateNodes(1.0,0.02,0.6,51,2,31,&nNodes,&nodes3D); // the length of x, y, z and the number of nodes in x, y, z.

   migrateNodes(1.5,0.99,1.0,nNodes,nodes3D); // migrate all the nodes with distance in x, y, z direction.

   The code will generate two important files: "test.bou" & "test.node". "test.bou" store the fixed nodes of the model used for Vega FEM solver. Rename "test.bou" into "mesh.bou" for Vega FEM solver.

2. Generate tetrahedral mesh from "test.node"
   Enter the command "tetgen.exe test.node" in windows command shell to get the tetrahedral mesh file "test.1.veg". Rename "test.1.veg" into "mesh.veg" for Vega FEM solver.
3. Generate membrane mesh for IBM solver.
   Run the code below:
   "FSI_Chengjun\LauderMembraneVibration\inputFiles_v5_membrane_iteration_forceCorrection**generateUnstruc_vegaSurfaceMesh_forMembrane.c**"
   Obtain "unstruck_surface_in.dat" for IBM solver and "vegaSurfaceMesh.dat" for Vega FEM solver.