

## Step-by-Step Structural Modes:

- 1) Create 2 folders. 1 to solve fluids, one to solve solid.

Start with the Solid Folder to extract Eigenmodes

- 2) In solid folder, make sure to have the following files

(1) Executables / config files

- gmsh2inp
- gmsh2inp.cc
- gmsh2inp.config
- a.out
- ccx\_2.19
- fd2vtk

(2) Solid mesh and config

- solid.geo
- solid.msh (create using gmsh-3)
- simflow.config

- 3) Start by creating the solid domain mesh in gmsh. Select fixed surfaces as nodes physical surface and the volume as a physical volume.

Let problem name = "solid"; physical volume = "panel"  
physical surfaces = "root"

2e) Create simflow.config file for simGmsh Cvt.  
give it correct panel name and solid model lines:  
→ Only for solid conversion

solid\_coordinates\_file = solid.crd  
solid\_connectivity\_file = solid.panel.cnn  
solid\_model\_file = solid.1.raw, solid.2.raw ....

5) Use simGmsh Cvt to create crd, cnn, nbc, srf files

6) Edit gmsh2inp.config and change PROBLEM name and CNN file to:

PROBLEM solid 2 — problem name  
CNNFILE panel ← physical volume  
NELMNODES no. of nodes per element  
NSRFNODES no. of nodes per surface element  
NODES root

7) Run ./gmsh2inp. Note that the created .inp file has 3 numbers:

NODE 50500 lines — ①  
ELEMENT 44100 lines — ②  
NSET 100 lines

8) Copy reference code lines from an older inp file into the newly created one.

These lines include:

\* BOUNDARY (x3)  
Root, 1 (or 2, or 3)

\* NSET

↳ put in no. of nodes — (1)

\* ELSET

↳ put in no. of elements — (2)

\* MATERIAL, NAME = EL

\* ELASTIC

put young's modulus, put poisson ratio

\* DENSITY

⋮ Put Density  
⋮

\* END STEP

9) Run ./ccx - 2.19 problemname

.cvg, .dat, .eig, .fcd files get created.

10) Run ./fcd2vtk problemname no. of modes

(creates .raw and .vtk files for every eigenmode)

- 11) Create new folder for fluid solving.  
Create new geo/msh file for the fluid domain with cavity for solid.

Make sure to mesh solid surface same as the surface meshing done in solid mesh.

Create new simflow.config with the usual intent of solving fluid flow.

Let domain problem name be "domain"  
and the surface of the flexible object be "panel" which has nbc, str files.

- 12) Run sim Gmsh Cnut to gen domain.crd and domain.panel.nbc (and others for later)

- 13) Copy domain.crd and domain.panel.nbc back to the structural folder.

- 14) Change simflow.config in structure folder to include

# map into fluid coordinates, and nodes to deform

crd = domain.crd  
node\_file\_list = domain.panel.nbc

15) Run `simEigen -h` to check all values are correct.

<code>-pb</code>	<code>solid</code>
<code>-file</code>	<code>undefined</code>
<code>-schr</code>	<code>solid.ord</code>
<code>-scnn</code>	<code>solid.panel.cnn</code>
<code>-scndim</code>	<code>3</code>
<code>-smode</code>	<code>solid.1.haw, solid.2.haw, ...</code>
<code>-ord</code>	<code>domain.ord</code>
<code>-node</code>	<code>domain.panel.nbc</code>
<code>-srf</code>	<code>none</code>
<code>-ccf</code>	<code>none</code>
<code>-gimp</code>	<code>TRUE</code>
<code>-lbuff</code>	<code>FALSE</code>
<code>-v</code>	<code>2</code>

16) Once confirmed, `simEigen` is good to run.  
(Use Nihar's version. That works).

17) The output will contain a section where a snippet of `def` file needs to be copied:

```
FLEXIBLE_BODY ("solid") { }
```

Copy this and keep aside.  
`xmd`, `ymd`, `zmd` get created.

18) Go to fluid folder and copy into it the xnd, ynd, znd files created in structure folder.

19) Make a definition file with ALE framework.  
Give match mesh velocity at the surface being deformed.

20) Copy the output from SimEigen into this definition file.

Make the following changes in syntax:

FLEXIBLE\_BODY → flexibleSolid  
mesh\_displacement → meshDisplacement  
num\_sub\_steps → numSubSteps  
num\_modes → numModes  
surface\_outputs → surfaceOutputs

21) The simulation is ready to run.