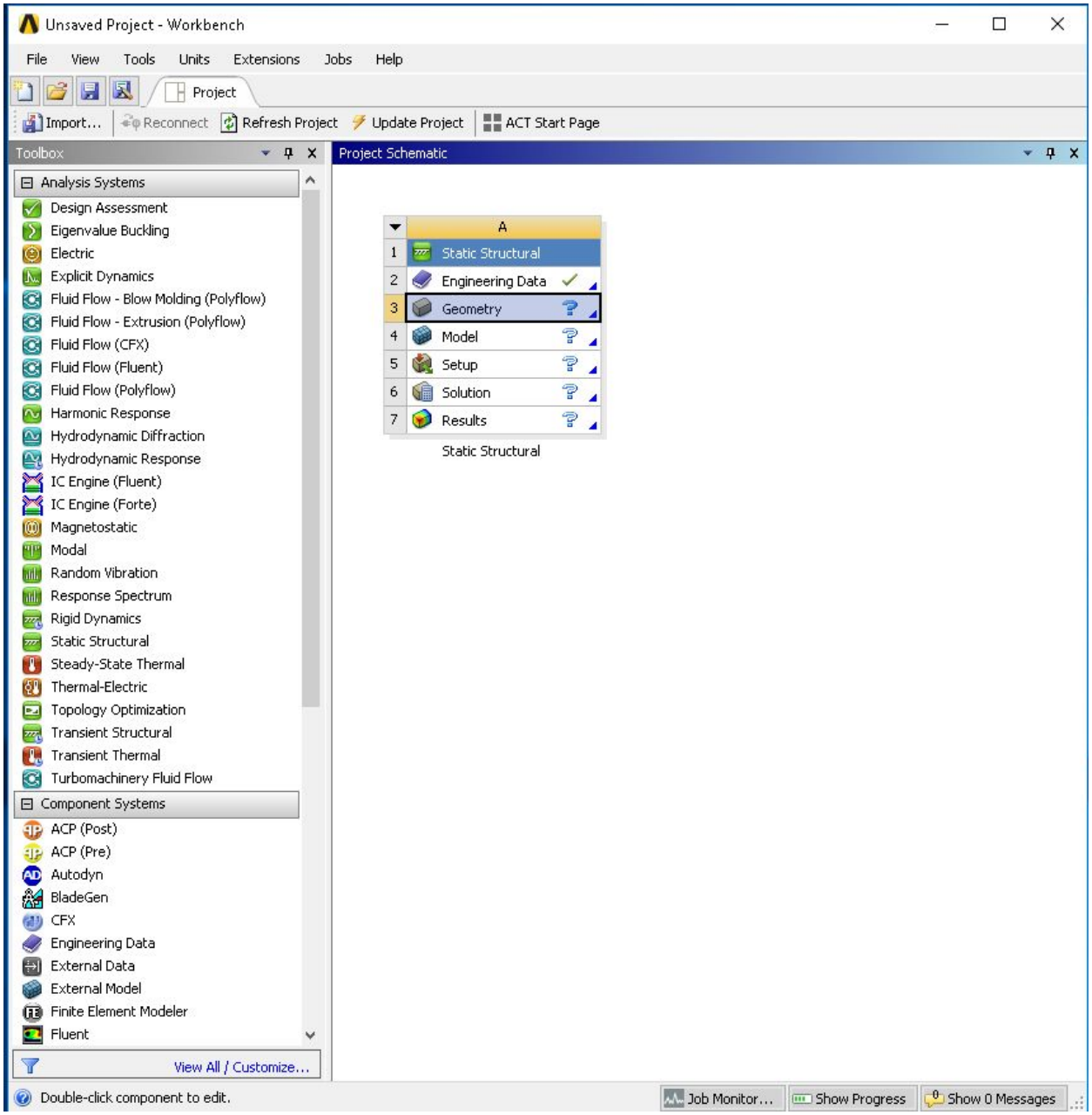
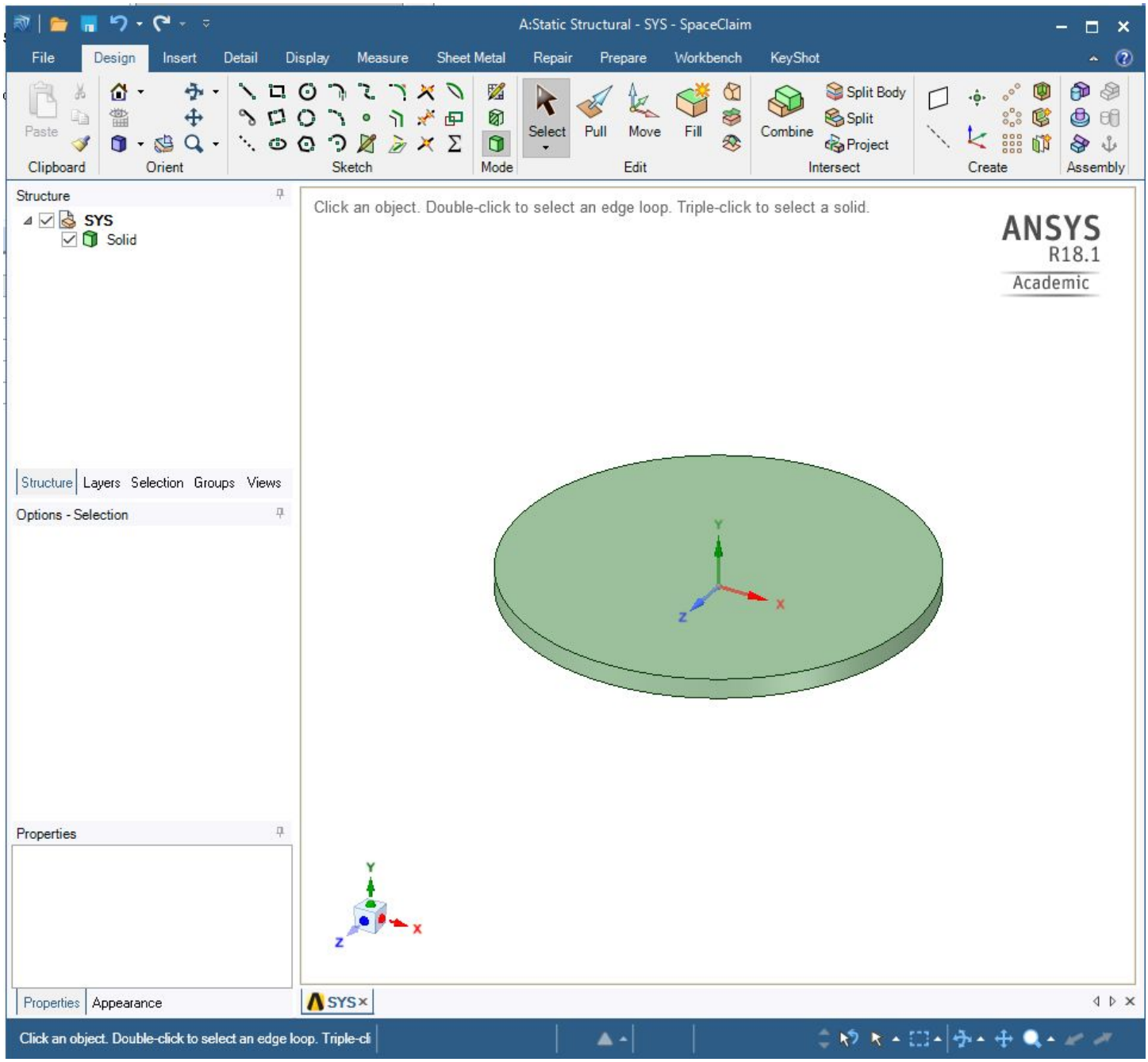
Using Forge to solve mechanical, static structural problems.
Ticket 1519497 - October 2017
Student - Garrett Finley
Ansys version 18.1

Summary

1. Open Ansys Workbench on a Windows system.
2. Using Static Structural Analysis System

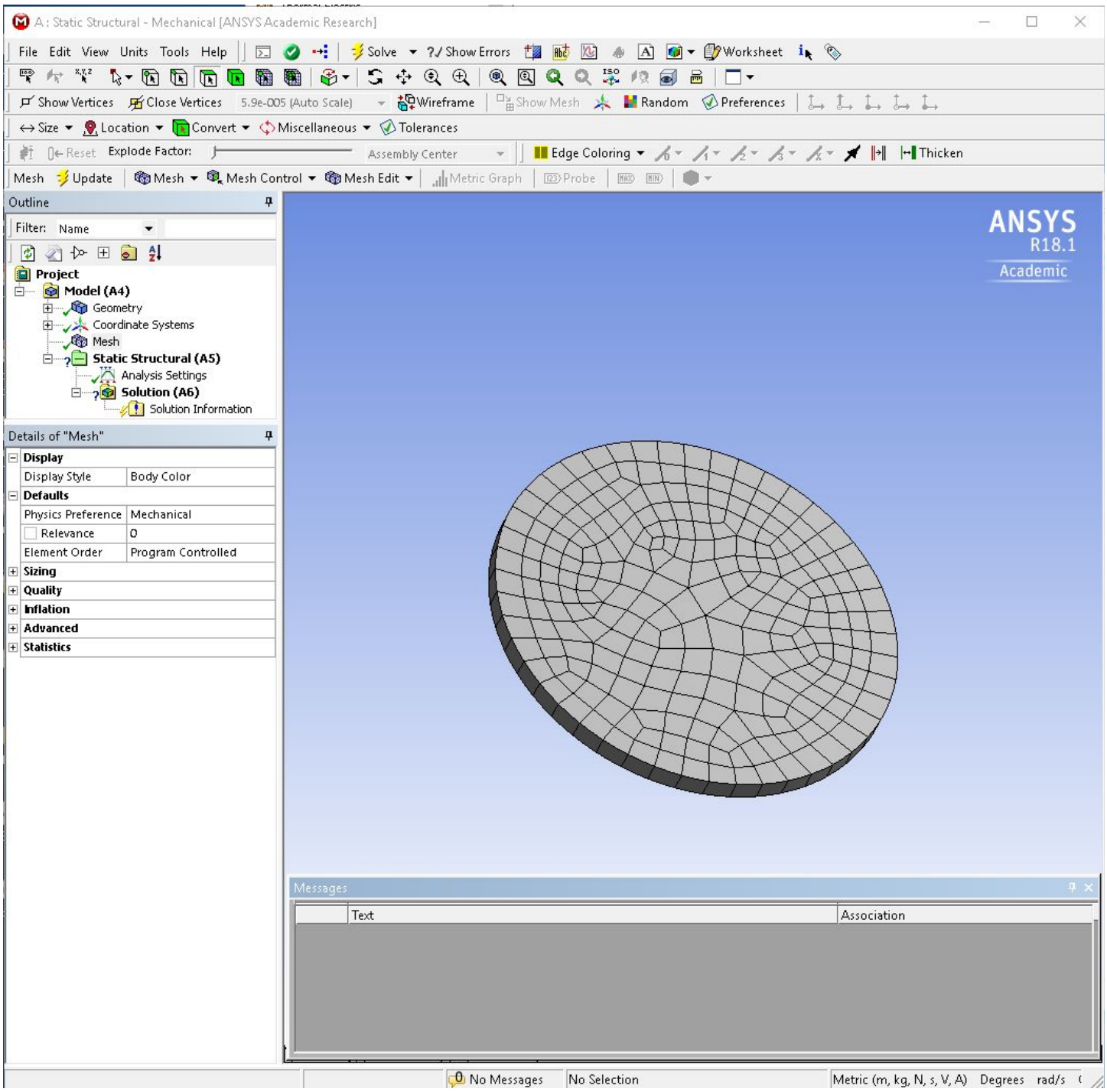


3. Double-click Geometry on the new element to open Ansys' SpaceClaim modeler

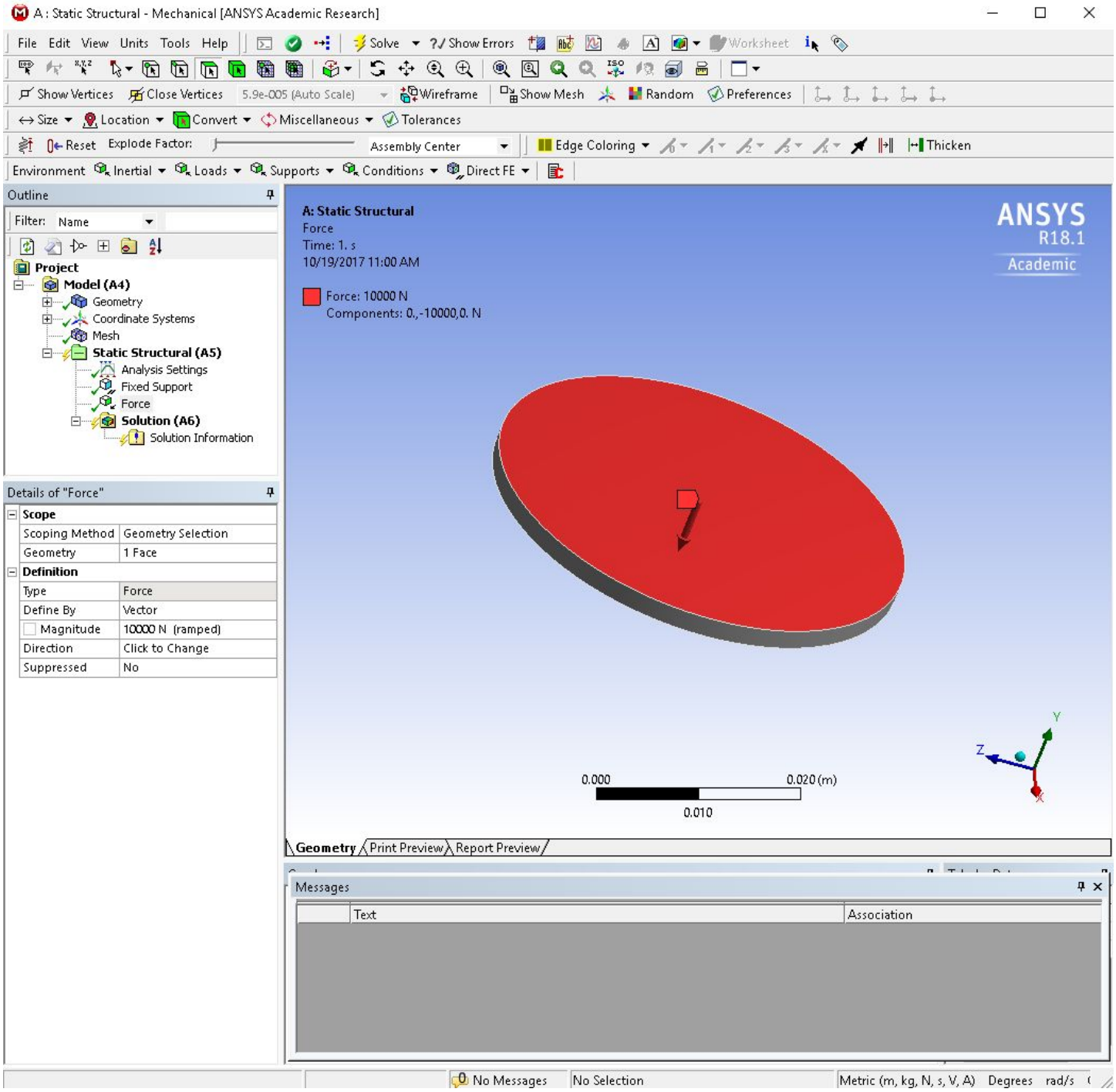


4. Build your geometry model and save the model to a new folder
5. Back on the workbench, click on the Model section - this opens the Mechanical GUI

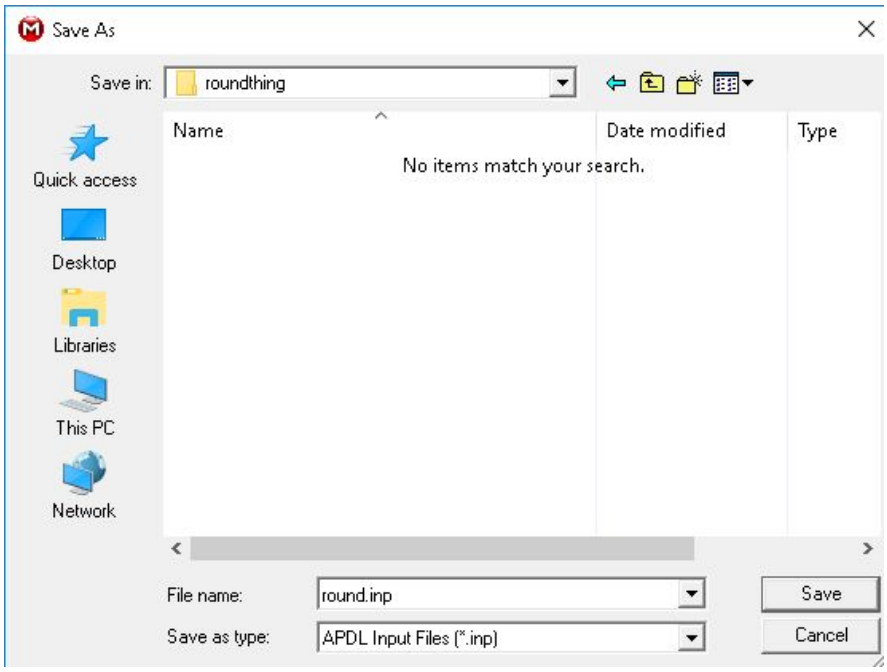
6. Click generate Mesh



7. Under Static Structural (A5), click a face to apply the load, supports, conditions, set those values, then direction.
8. In Solution (A6), select a parameter to determine its value, deformation, stress, strain, etc.



9. At this point, go to Tools, Write Input File…, the important .inp file and save it in the same project folder as the model/mesh.



10. This project folder needs to be copied to your Forge home folder. Use WinSCP or Filezilla, or some other SFTP program.

```
[rlhaffer@login-44-0 ANSYS]$ pwd
/home/rlhaffer/unittests/ANSYS
[rlhaffer@login-44-0 ANSYS]$ ls -al roundthing/
total 272
drwxrwxr-x  2 rlhaffer rlhaffer     54 Oct 19 11:03 .
drwxrwxr-x 12 rlhaffer rlhaffer   4096 Oct 19 11:03 ..
-rw-rw-r--  1 rlhaffer rlhaffer  33135 Oct 19 11:03 Design1.scdoc
-rw-rw-r--  1 rlhaffer rlhaffer 234021 Oct 19 11:03 round.inp
```

```
[rlhaffer@login-44-0 roundthing]$ ll
total 276
-rw-rw-r-- 1 rlhaffer rlhaffer  33135 Oct 19 11:03 Design1.scdoc
-rw-rw-r-- 1 rlhaffer rlhaffer 234021 Oct 19 11:03 round.inp
-rw-rw-r-- 1 rlhaffer rlhaffer     47 Oct 19 11:05 round.log
-rw-rw-r-- 1 rlhaffer rlhaffer    365 Oct 19 11:05 round.sbatch
```

11. Create a ansys .log file - for telling the ansys mechanical solver to do things
12.
```
/INPUT,'<inputfilename>','inp',,0,0
/SOLU
SOLVE
FINISH
```
13. SBATCH file
    a.
```
#!/bin/bash
#SBATCH --job-name=jobname
#SBATCH --ntasks=# of processors
#SBATCH --mem=amount of RAM
#SBATCH -p=requeue
#SBATCH --time=00:60:00
--comment=#SBATCH --mail-type=begin
--comment=#SBATCH --mail-type=end
#SBATCH --export=all
#SBATCH --out=Forge-%j.out
```

```
module load ansys/18.1

time ansys181 -j jobname -b -dis -np ## < /home/<userID>/Ansysprojectfolder/ansysproject.log
```

b. Set the number of processors, the amount of RAM in MBs, the forge queue of your choice...free/requeue is typical, time...this is subjective

c. Load the ansys module corresponding to the version you used when creating the model - to avoid compatibility issues.

d. Set the -np (# of processors) in the ansys script, to the number set in the sbatch command

14. Run the sbatch command
15. Many files and folders will be generated during the simulation time
16. Import file is the .rst (results) file.

```
-rw-rw-r-- 1 rlhaffer rlhaffer  851968 Oct 19 11:19 round.rst
```

17. Copy the .rst file back to a system with the Ansys workbench installed, then load the workbench project, the mechanical gui to look at the design, then under Tools, Read Results file…
18. This will prompt to import the .rst file, to show the solved simulation details.