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

🔞 A : Static Structural - Mechanical [ANSYS Acad	demic Research]	- 🗆 🗙
 File Edit View Units Tools Help 🖂 🕻	2 🕶 🚽 Solve 🔻 🦅 Show Errors 🎁 📷 🔞 🥢 🧥 \Lambda 🞯 🖛 🕼 Worksheet in 🗞	
Show Vertices Har Close Vertices 5.9e-005	$[Auto Scale] = \bigcirc $	
] ↔ Size ▼ 🧖 Location ▼ 🖪 Convert ▼ 🗘 M	liscellaneous 🔻 🐼 Tolerances	
] 🙀 ()← Reset Explode Factor:)	Assembly Center 👻 📗 Edge Coloring 👻 ゎ 🍡 🖓 🗸 🖉 🖓 👘 🖬 🖛 Thicken	
🛛 Mesh 孝 Update 🚽 🆚 Mesh 👻 🔍 Mesh Cont	rol 🔻 🏟 Mesh Edit 👻 📊 Metric Graph 🛛 🖾 Probe 🛛 🛲 💷 💷	
Outline 7		1.0000000000000000000000000000000000000
Filter: Name 👻		ANSYS
] 🔁 🖉 🐎 🖽 🗟 🛔		K10.1
 Project Model (A4) Geometry Coordinate Systems Static Structural (A5) Analysis Settings 2 Solution (A6) Solution Information 		Academic
Details of "Mesh" 7		
- Display		
Display Style Body Color	XIIII	
- Defaults	XXXXTTXXX	
Physics Preference Mechanical	XXXXHTHXXX	
Relevance 0	AXXXXXITTTYXXXA	
Element Order Program Controlled	LANN XETTINXX	
+ Sizing		
+ Quality		
+ Inflation	THEXXXXXXIII	
+ Advanced	THXXXIIII	
Statistics		
	Messages	# ×
	Text	
	No Messages No Selection Metric (m, kg, N, s, V, A)	Degrees rad/s (/

- 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.

🙆 Save As				×
Save in:	roundthing	•	🗢 🗈 💣 📰 🔻	
Quick access	Name	^ No items match your	Date modified search.	Туре
Desktop				
Libraries				
Network				
	۲.			>
	File name:	round.inp	•	Save
	Save as type:	APDL Input Files (*.inp)	.	Cancel

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

total 272	Lag	LN-44-0 AF	isrsja is	-at ro	Indtr	1111	97	
drwxrwxr-x	2	rlhaffer	rlhaffer	54	0ct	19	11:03	
drwxrwxr-x	12	rlhaffer	rlhaffer	4096	Oct	19	11:03	
- FW- FW- F	1	rlhaffer	rlhaffer	33135	Oct	19	11:03	Design1.scdoc
- FW- FW- F	1	rlhaffer	rlhaffer	234021	Oct	19	11:03	round.inp

total 276								
- FW- FW- F	1	rlhaffer	rlhaffer	33135	0ct	19	11:03	Design1.scdoo
- FW- FW- F	1	rlhaffer	rlhaffer	234021	0ct	19	11:03	round.inp
- FW-FW-F	1	rlhaffer	rlhaffer	47	Oct	19	11:05	round.log
- DV - DV - F	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 --mem=amount of RAM
 #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.

