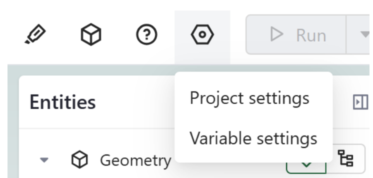
Importing/Starting a New File:

For each tab angle, we will have a different CFD project file

1. Go to Zephryus Controls workspace is Flow360
2. Create new project file, green button on top right
3. Update project name to be: 10_deg_control_surface (replace 10 with correct tab angle)
4. Release set to 25.7
5. Units set to inch
6. Description: All mach numbers for 10 degree tab angle (replace 10 with correct tab angle)
7. Creation method from geometry
8. Select STEP file from folder and import
    a. Folder: \MIT Rocket Team - Zephyrus\AeroD\CFD\Flow360 CFD control surface\CAD_Degrees

Setting up your workspace:



1. Hit project settings in top right
2. Turn on geometry mesher and beta mesher
3. Make sure release 25.7
4. Rename to 10 degree (insert your angle)
5. Go into operating conditions on left taskbar
6. Type = mach (in velocity dropdown)
7. Start with lowest mach number for first run (this will change each run you perform on this geometry
8. Plug in the density and temperature according to CFD MAtrix spreadsheet that corresponds to your desired mach and tab angle
9. Hit mesh parameters
10. Surface edge growth = 1.2
11. Max edge length = 10mm
12. Surface aspect ratio = 10
13. Geo accuracy = .5mm
14. Curvature res angle = 12 deg
15. Boundary layer growth rate = 1.2
16. Boundary layer 1st layer thickness = 1e-6m

17. Go to time on left, then steady, max steps = 5000
18. Go to outputs list, scroll down, set output fields (default should be fine, add more if needed)
19. Go to the reference dimensions in the left bar, and set the proper reference dimension for what you want to measure
20. Go to right bar, hit add button next to slices, create new slice named “center”, leave default params
21. Add another slice, adjust the **y** to be -8.75 inches, named control surface and normal to be x: 0, y:1, z:0
22. Go to left task bar
23. Hit plus sign next to output list and add slice
24. Select control surface slice, set output parameters to Cp, T, and velocity (non-dimensional), name slice control surface
25. Repeat for center slice
26. Run sim in top right
    a. Note first run will take a while

Adding Additional Mach Numbers

1. After first run, you can fork this workspace to put in different operating conditions
2. Click on project file in top task bar
3. This will open a work tree
4. click on you flow run and hit fork
5. Rename to be the degree and mach number you are running
6. Adjust operating conditions for new mach number (can follow previous steps)
7. Follow steps 18-25 to set up the correct slices

** Note: when a run is done, highlight it green in the CFD matrix spreadsheet