11/1/23 - How to set up CFD

Wednesday, November 1, 2023   11:10 AM

Setup for CFD:
1. Import geometry
2. Generate mesh
3. Create named selections
   a. Symmetry
   b. Walls
   c. Inlet
   d. Outlet
   e. Chamber: hide domain geometry, face selection, box select chamber
4. Check mesh
5. Quality mesh check (orthogonal quality should be above 0.1)
6. General
   a. Steady time
7. Models
   a. Energy equation on
   b. Transition SST (combines k epsilon and k omega)
8. Materials
   a. Fluid: air -> constant density
9. Boundary conditions
   a. Inlet: velocity = 2m/s, 5m/s, 10m/s, thermal = 263.15K (-10C average air temp in deployment area)
   b. Outlet: pressure = 0Pa, thermal = 263.15k
   c. Walls: moving wall -> speed = 2m/s, 5m/s, 10m/s, x-direction=1, no-slip
10. Methods: SIMPLEC
11. Report definitions
    a. New force report: force -> zones: chamber, report file, report plot
    b. New flux report: mass flow rate -> zones: inlet and outlet, report file, report plot (t determine steady state)
12. Initialization
    a. Standard
    b. Gauge pressure: 0Pa
    c. X velocity: 2m/s, 5m/s, 10m/s
13. Run calculation
    a. 500 iterations

a. Standard
        b. Gauge pressure: 0Pa

13. Run calculation
        a. 500 iterations