

Exercise 5

Simulation of non-premixed combustion in jet engine

Purpose

To simulate the evaporation and combustion of kerosene in the jet engine.

Duration: 2h

Mesh

Figure 1 shows the computational mesh for simulation of the flow and combustion in the jet engine. The length of the jet might be set to 1-2m and $D=0.48\text{m}$ (see fig. 1). The other dimensions are arbitrary.

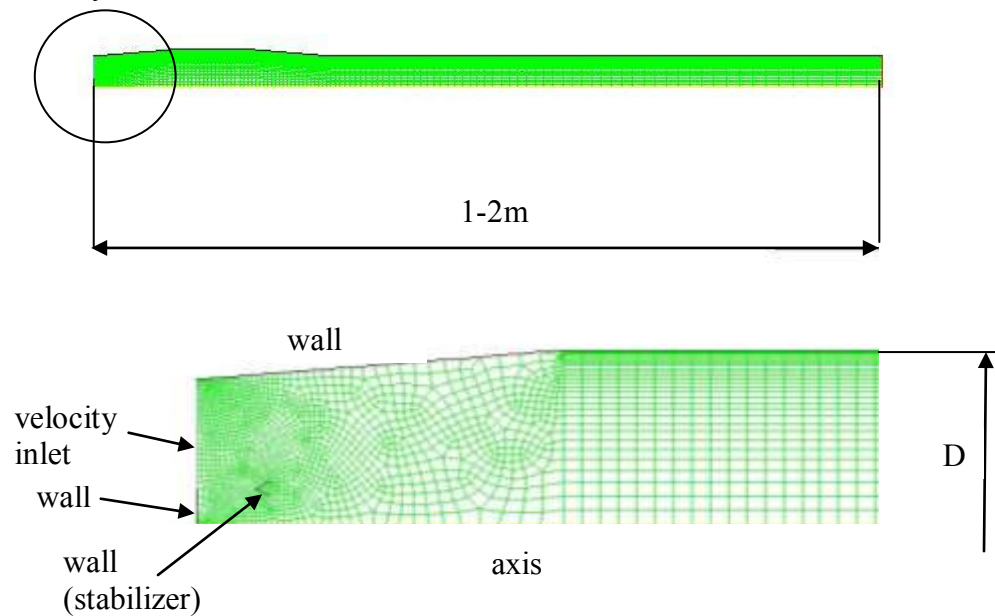


Fig.1. Computational mesh.

Fluent (settings for Fluent ver 12)

1. Single precision solver. **Define/General:** pressure based, steady, axisymmetric swirl.
2. **Define/Models/Energy.** Activate solution of the energy equation.
3. **Define/Models/Viscous.** Activate the k- ϵ turbulence model.
4. **Define/Models/Species.** Activate the non-premixed combustion.
 - a. Switch to **Boundary** under **PDF Table creation** and enter c12h23 (kerosene) under **Boundary Species**. Press the **Add** button to activate the settings. The fuel 'c12h23' will be added.
 - b. Set the mass fraction for 'c12h23' to 1 for fuel and 0 for oxidizer (oxid) boundary.
 - c. Activate a preparation of the PDF chemistry table under **Table**. One can use **Display PDF Table** to see the variation of temperature, mixture density etc as a function of mean mixture fraction and mixture fraction variance. Fluent displays the maximum temperature for the mean mixture fraction of about 0.06 (stoichiometric mixture).
5. **Define/Models/Discrete Phase.** Define the discrete phase model. The interaction with the continuous phase should be activated.
6. The kerosene droplets have to be injected in the vicinity of 'stabilizer'. One has to know the injection point (radius for axisymmetric problem considered here). Do this using **Surface/Point/Select Point with Mouse**. The combustion will be initialized by specifying the high temperature on the surface called 'stabilizer'. The air temperature will be raised locally in the vicinity of 'stabilizer'. As result, the kerosene droplets will start to evaporate.
7. **Define/Injections.** Enter Group under **Injection Type**. Specify about 10-20 particle streams. Select Droplet under **Particle Type** and kerosene-liquid under **Material**. If the kerosene-liquid is not on the list, it should be added in **Define/Materials/Droplet Particle**. Enter c12h23 under **Evaporating Species**. Provide the coordinates of the First and Last point (the same). Define the velocity of the droplet streams X-Velocity=20-30m/s, Y-velocity 1-5m/s, Z-velocity=0. The velocities can be different for the first and last point. Droplet diameter: $D=1e-5m$, temperature: $T=300K$, mass flux: $0.002kg/s$.
8. **Define/Boundary Conditions**
 - a. Velocity inlet: Air velocity: x-velocity: 5m/s, swirl velocity: first select 0m/s. Next, increase the swirl velocity in order to stabilize the combustion process. Temperature 300K.
 - b. Stabilizer-wall. Set the temperature to 5000K (fictitious value).
9. Initialize the solution from the velocity inlet.
10. Run the calculations for next 20-30 iterations with high temperature specified on the 'stabilizer' surface. The high temperature is used to mimic the ignition process (spark). If the combustion process is activated (high temperature about 2000 K), change the type of the boundary condition on stabilizer surface from temperature to heat flux. Set the heat flux to : $0W/m^2$ and finish the computations. Residuals should decrease below $1e-4$.

11. Make some additional simulations with different swirl velocity at the velocity inlet. Do not initialize the solution for gas phase. Rise the swirl velocity up to 5 m/s. Compare the results.
12. Analyze the influence of the swirl velocity on the flame location and on the temperature inside the jet engine. Use the contour plots or the xy-plot to visualize the temperature. What is the role of swirl velocity (stabilizing or destabilizing the combustion) ?