# CFD Model Setup

## Boundary conditions

In this test case the experiments were performed seperately to measure flow and mixture field data. In order to compare the flow field data, obtained from simulations, with the experimental data air is used at both fuel jet and co-flow inlets with velocities 61 & 20 m/s respectively. For the simulations carried out to measure mixture field data different fuels with different velocities, as mentioned in the literature [6], have been used at the fuel jet and pure air at co-flow inlet.

### Boundary conditions (Flow field):

Inlet velocities and turbulent inlet boundary consitions have been utilised as per specified in the literature [6]. Air at 25°C with constant properties was used as fuel at both jet & co-flow inlets.

|  |  |  |
| --- | --- | --- |
| Name | ANSYS CFX 16.1 | ANSYS Fluent 17.0 |
| Co-flow inlet | Normal Velocity = 20 m/s  Turbulent Intensity = 2%, Eddy Viscosity Ratio = 10 | Normal Velocity = 20 m/s  Turbulent Intensity = 2%, Eddy Viscosity Ratio = 10 |
| Jet inlet | Normal Velocity = 61 m/s  Turbulent Intensity = 5%, Eddy Viscosity Ratio = 10 | Normal Velocity = 61 m/s  Turbulent Intensity = 5%, Eddy Viscosity Ratio = 10 |
| Outlet | Relative Pressure = 0 Pa | Gauge Pressure = 0 Pa |
| Bluff body | No slip wall | No slip wall |
| Fuel pipe | No slip wall | No slip wall |
| Wall 1& 2 | No slip wall | No slip wall |
| Symmetry 1&2 | Symmetry | Symmetry |

Table 9

### Boundary conditions (Mixing field):

The boundary conidtions for the mixing field except the jet inlet will remain the same as shown in the Table 9

|  |  |  |
| --- | --- | --- |
| Name | ANSYS CFX 16.1 | ANSYS Fluent 17.0 |
| Jet inlet | Normal Velocity = 50\* m/s  Turbulent Intensity = 5%, Eddy Viscosity Ratio = 10 | Normal Velocity = 50\* m/s  Turbulent Intensity = 5%, Eddy Viscosity Ratio = 10 |

Table 10

\*Note : The velocity at the fuel jet inlet will vary depending on the experiment performed as mentioned in the literature [6].

## Material Properties:

### ANSYS CFX 16.1

Flow field:

Air at 25°C with constant properties was used at both fuel jet & co-flow inlets.

Mixing field:

For the mixing field measurement, three different mixtures were prepared:

1. Mixture Air was defined as a Fixed composition mixture[12] of Ar(1.28%), O2 (23.15%), N2 (75.52%) and CO2 (0.0035%).
2. Mixture Fuel was defined as a Fixed composition mixture of CNG, Ethylene and LPG. Composition of which can be found in the literature [6].
3. Air-Fuel Mixture was defined as a Variable composition mixture[12] of Mixture Air and Mixture Fuel, where mixture air was defined as constraint.

The composition of the fuels, used at the fuel jet inlet, are used as per mentioned in the literature[6].The purpose for using three different mixtures is to avoid the calculation of the additional transport equations for all the species. Hence only one additional equation will be solved for the entire mixture. Mass fraction of the fuel is defined as 1, at fuel inlet, and zero at co-flow inlet.

### ANSYS Fluent 17.0:

Material used for the Flow Field calculations was same as in ANSYS CFX 16.1

Mixing Field:

Here only one mixture was made with all the component species of fuel and air respectively. Care

was taken to have the same material properties i.e. density, dynamic viscosity etc for species in ANSYS CFX and Fluent.

## Initialization:

### ANSYS CFX 16.1

In ANSYS CFX the initialization has been done by using the global initialization option. The domain is initially filled with air. The initial velocity field is provided with cartesian velocity components, as u = 20 m/s, v & w = 0. Initial turbulence conditions are chosen of type medium, turbulent intensity = 5% and Eddy viscosity ratio = 10. Mixture fraction of fuel is defined as zero here.

### ANSYS Fluent 17.0

Initialization is done using the hybrid initialisation option available in ANSYS Fluent. In some cases the initilization was done using the interpolation of the results obtained from the coarser mesh on to the next finer mesh, to achieve converged solutions. See Interpolation option in [13].

## Numerical Settings:

### ANSYS CFX 16.1

The convergence target was set to type MAX Residuals < 10-4 and the conservation target was set to 5.10-4. The discretization scheme used was high resolution [12] and the turbulent numerics scheme was also set to high resolution [12]. Auto timescale was the option used for the time step control. Maximum number of iterations was set to a value of 500\*.

### ANSYS Fluent 17.0

Flow analysis chosen here is of steady state type with pressure-based [13] solver. In all simulations Coupled method [13] is chosen for the pressure-velocity coupling. The gradient approximation is done by Least Squares Cell Based method [13]. For pressure second order method has been used. For momentum second order upwind scheme is used. Turbulent numerics have been approximated by using the First order upwind scheme. For all the species transport again a second order upwind scheme is used. Pseudo transient and higher order term relaxation [13] have also been selected. The residual type RMS (Root mean square) with a target value of 10-6 is selected for each solution variable. Time step method was chosen to be automatic with 800\*\* as the maximum number of iterations for all the simulations

\*, \*\* :In some cases to achieve converged results, for both the solvers ANSYS CFX 16.1 and ANSYS Fluent 17.0, the maximum number of iterations were increased and time step was reduced to a smaller value.