Numerical Investigation of a BFR using FLUENT

This report discusses about the attempt to reproduce the results which was published on the reacting flow inside a Burner Flow Reactor (Aalborg University) using commercial CFD code FLUENT. The previous work was concerned with the swirling turbulent cold flow and the flow with combustion of air with methane as a fuel. There were results for two swirler angles of 9.5˚ and 15.5˚. In the current simulations results were only reported for 9.5˚ swirler angle with and without combustion.

<u>Computational Mesh and Boundary Conditions:</u>

Figure 1: BFR Geometry with Mesh.

The geometry and mesh was created as per the dimensions given in the report and is shown in Figure 1. The geometry was meshed with quadrilateral cells and the element count was 36320. The mesh independency check was not performed but being worked on similar projects it is possible to settle that this mesh will give satisfactory results.

The inlet was specified as velocity inlet using BOUNDARY PROFILE FILE, the outlet was specified as pressure outlet, centre line was specified as axis and all the other entities as wall. The axial, radial and tangential (swirl) velocities which are specified at the inlet are plotted in Figure 2a, 2b and 2c. The velocity profile at the inlet is taken from the boundary conditions and the table given in the original report. The k-ε turbulence equation with standard wall functions was used and the value for k and ε was provided as 0.0375 and 0.15 at the inlet.

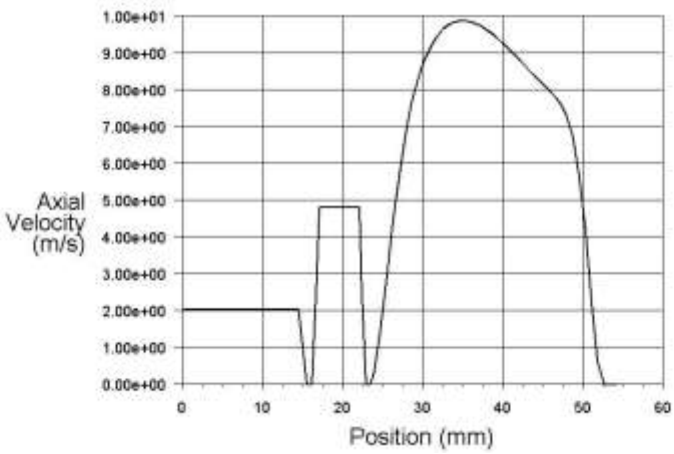

Figure 2a: Axial Velocity variation at the inlet

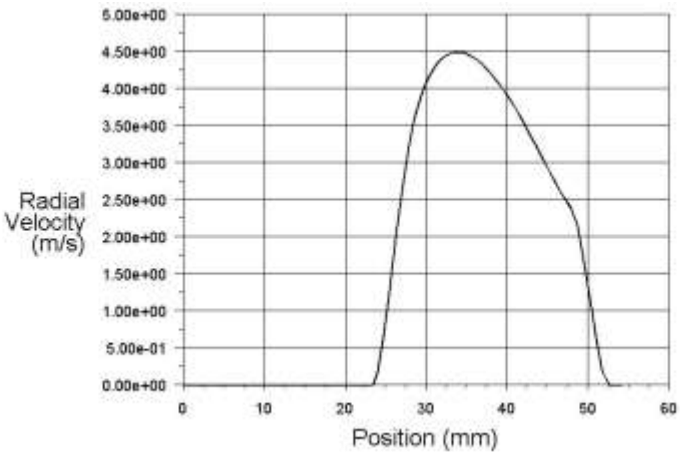

Figure 2b: Radial Velocity variation at the inlet

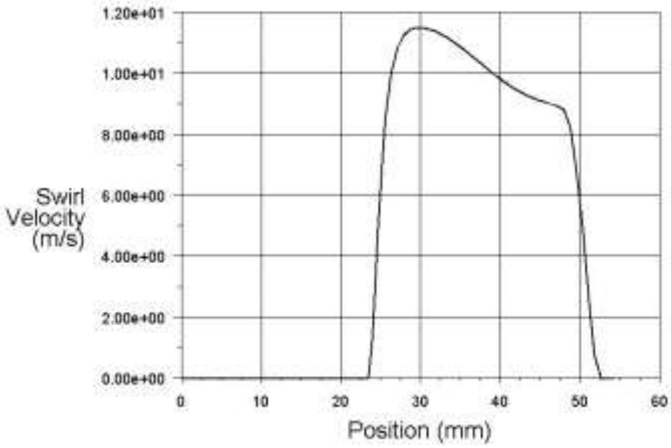

Figure 2c: Tangential (Swirl) Velocity variation at the inlet

<u>Results for Cold Flow Simulations:</u>

The velocity vector plot for the 9.5˚ swirler configuration is given in Figure 3. It is comparable with the results which are given in the report.

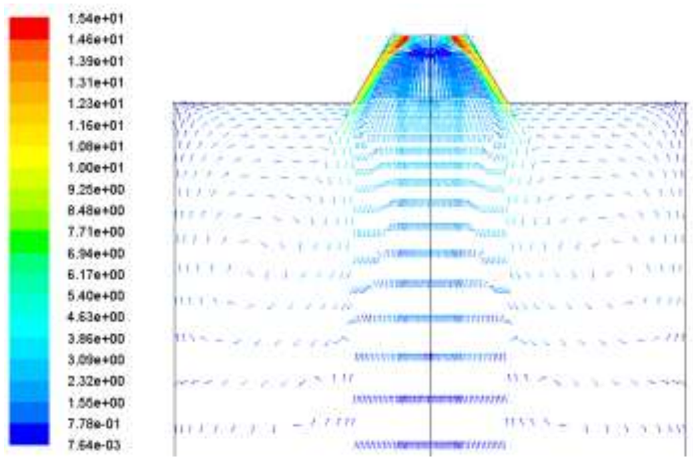

Figure 3: Velocity vector plot for 9.5˚ swirler configuration

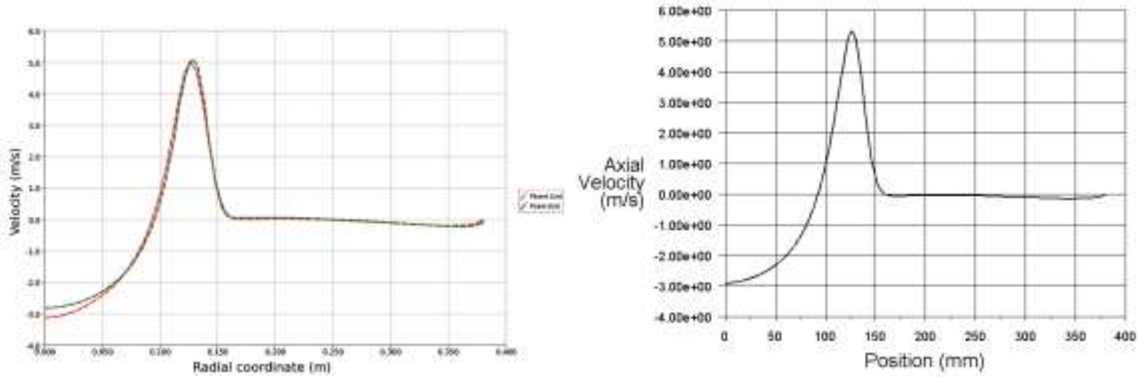

Figure 4a: Axial Velocity variation at x = 0.15 m

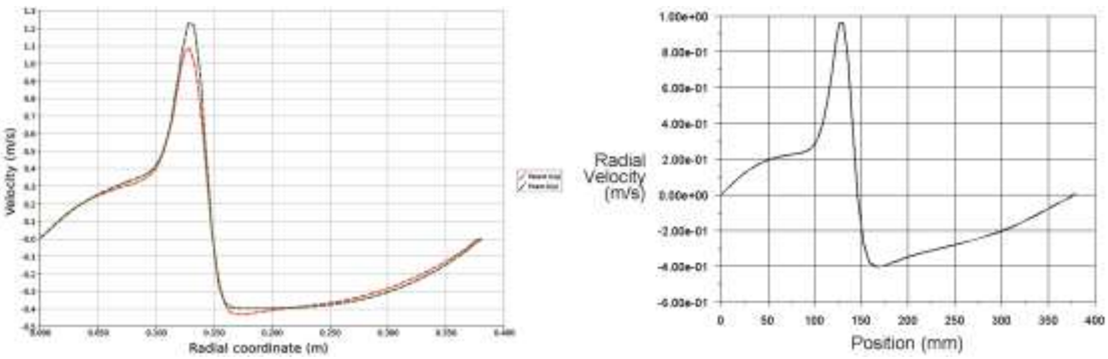

Figure 4b: Radial Velocity variation x = 0.15 m

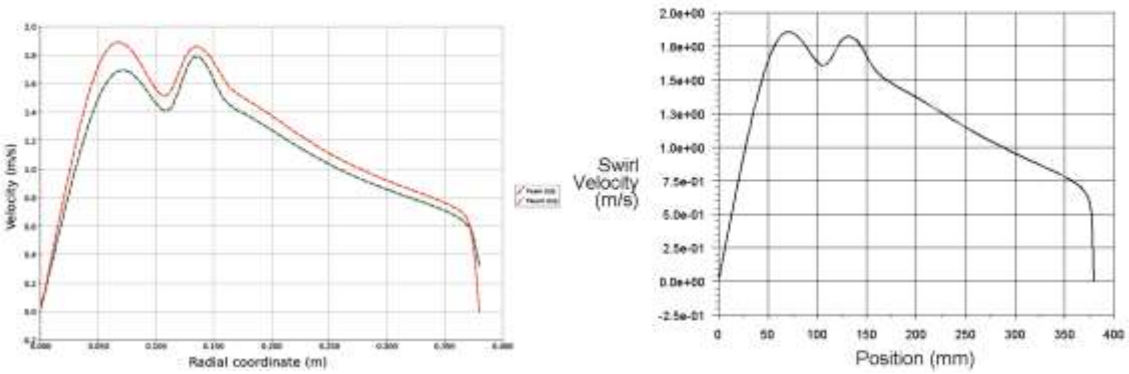

Figure 4c: Tangential (Swirl) Velocity variation at x = 0.15 m

The Figures 4a, 4b and 4c show the velocity variation in axial, radial and tangential direction at 0.15 m from in the inlet plane. The left hand side figure refers to velocity variation given in the report and the right hand side figure refers to the current FLUENT simulation results. From the above figures we can conclude that the overall trend is reproduced but the maximum values are slightly different from the report, particularly the maximum axial and radial velocities. Having simulated the cold flow simulation with desired results, the combustion simulation was performed as per the boundary conditions given in the report.

FUEL RICH SIMULATIONS:

To perform the fuel rich combustion simulation, the cold flow results were used as initial conditions and the whole domain was patched with 2000K to ignite the fuel and oxygen mixtures. The residuals were in the order of 1e-6, when the combustion simulation was independent with the number of iterations. The temperature, CH_4 and CO_2 contours were given in Figures 5a, 5b and 5c. The maximum temperature in the current simulation is 2066K, whereas the maximum temperature given in the report was 2130K and the relative error is about 3%. However the temperature contour and the other mass fraction contours were comparable with the published results in the report.

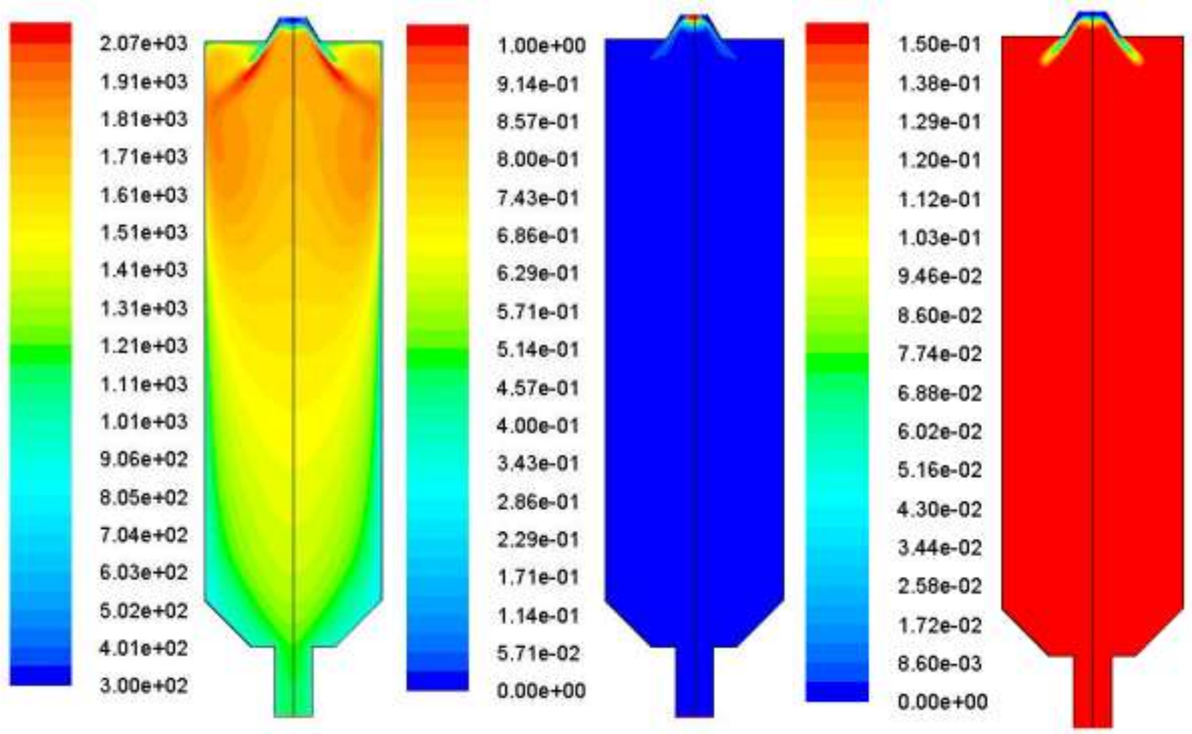

Figure 5(a) Temperature Contours 5(b) CH_4 Contours 5(c) CO_2 Contours

Species	Calculated	Fluent	Reacting Foam	Fluent (Current Simulation)
CO_2	0.1239	0.1503	0.1499	0.1499
H_2O	0.0943	0.1231	0.1228	0.1227
CH_4	0.0675	0.0128	0.0133	0.0139
N_2	0.7141	0.7138	0.714	0.7135

Table 1: Numerical values of mass fraction at the outlet (Fuel rich conditions)

The mass fractions of the various species were calculated using area weighted average at the outlet and given in Table 1. It can be observed that the difference in the report and the current FLUENT simulations were relatively small except the value for methane. The current simulation reports more methane compared with older result at the exit which means that there is more un-burnt fuel at the exit which accounts for the reduced maximum temperature.

FUEL LEAN SIMULATIONS:

To perform the fuel lean combustion simulation, the cold flow was simulated first and the steps followed are as explained earlier for fuel rich combustion. The temperature, O_2 and CO_2 contours were given in Figures 6a, 6b and 6c. The maximum temperature in the current simulation is 2251K, whereas the maximum temperature given in the report was 2200K and the relative error is about 2.3%. The temperature contour is comparable but the other mass fraction contours were slightly different from the published results in the report.

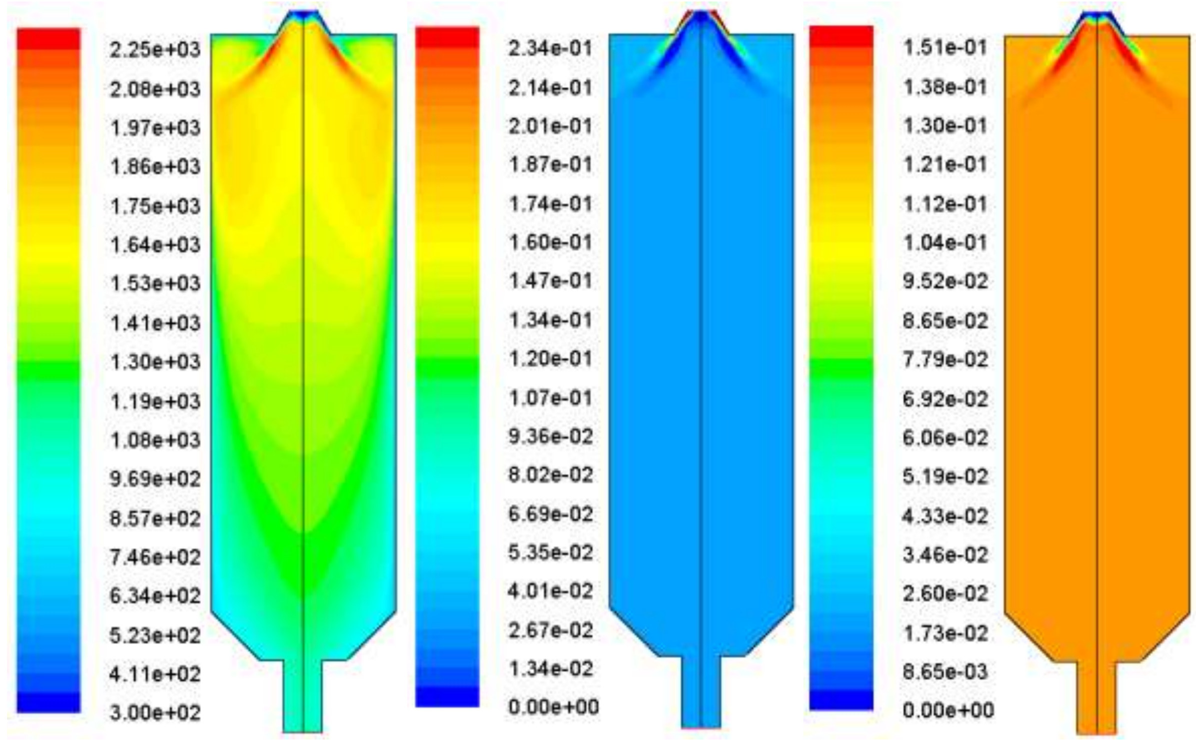

Figure 6(a) Temperature Contours 6(b) O_2 Contours 6(c) CO_2 Contours

Species	Calculated	Fluent	Reacting Foam	Fluent (Current Simulation)
CO2	0.1297	0.1314	0.1403	0.1328
H2O	0.1061	0.1075	0.1148	0.1088
O2	0.0334	0.0318	0.0188	0.0297
N2	0.7308	0.7292	0.726	0.7287

Table 2: Numerical values of mass fraction at the outlet (Fuel lean conditions)

The area weighted average mass fractions of the various species were calculated at the outlet and given in Table 2. It can be observed that the difference in the report and the current FLUENT simulations were relatively small except the value for oxygen. The current simulation reports less oxygen compared with older result at the exit which means that there is less oxidizer at the exit which accounts for the higher maximum temperature.

CONCLUSION:

The current simulation results were comparable with the results which were given in the report. The slight divergences from the results and their consequences were explained with the maximum temperature in the domain and mass fractions of species which were calculated at the outlet. The difference in results might depend strongly on the variations at the inlet and it might change a smaller amount depending on the mesh count.