



Technische
Universität
Braunschweig



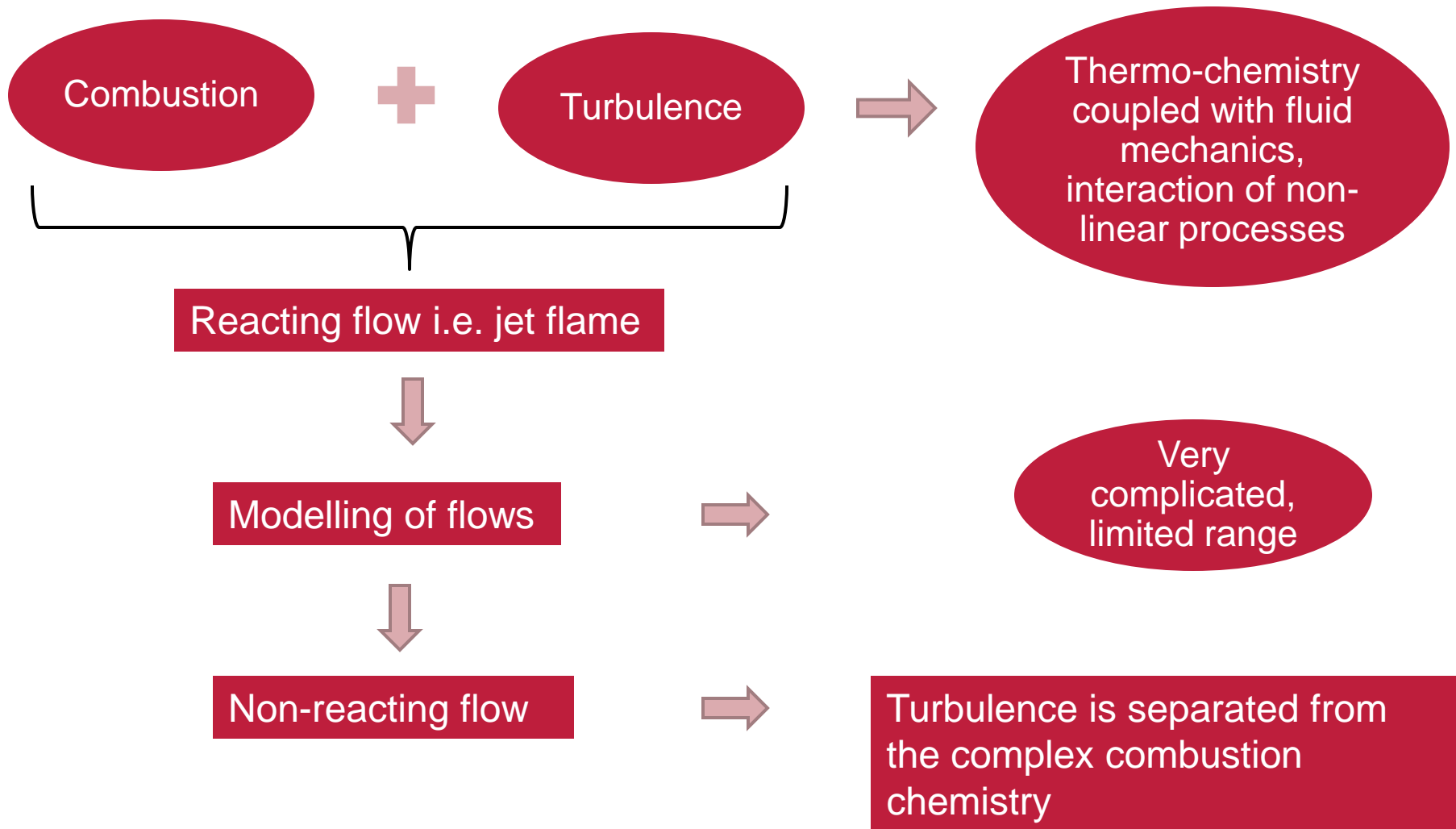
Investigation of two CFD validation test cases for turbulent mixing of different gaseous species (fuel and oxidizer) in typical laboratory flame configurations

- Student : Malav M Soni
 - Supervisor :
- Date : 12/10/2015

Content

- Objectives of present work
- Turbulence and its theory
- Turbulence models (RANS) used for investigation
- Test case I ([Sandia National Laboratory Propane Jet](#))
 - Experimental Data
 - Computational Domain
 - Quality Criteria
 - Conclusion
- Test case II ([Sydney Bluff Body Jet](#))
 - Experimental Data
 - Computational Domain
 - Quality Criteria
 - Conclusion

Objectives of the present work



Objectives of the present work

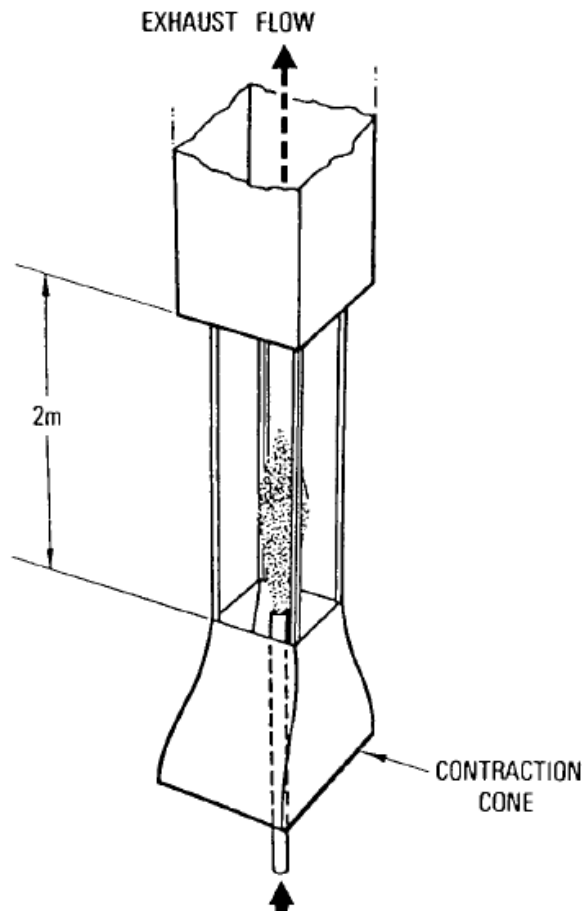
- **Aim** :
 - To study and investigate the flow structure and the turbulent mixing behind the bluff body, in the absence of complex combustion chemistry and heat release i.e. isothermal, with different RANS models
 - **ANSYS CFX 16.1** and **ANSYS Fluent 17.0**
- This study is divided in to parts:
 - **Part I** : Investigation of the Sandia National Laboratory Propane Jet (**quasi-2d**)
 - **Part II** : Investigation of Sydney Bluff Body Jet Flame (**full 3d**)

Part I – Sandia National Laboratory Propane Jet

- Experimental Reference
 - Robert W. Schefer “Data Base for a Turbulent, Non-premixed, Non-reacting Propane-Jet Flow “, Sandia National Laboratories Research Report, 1985, pp. 1-21.
 - R.W.Schefer and R.W.Dibble,”Mixture Fraction Field in a Turbulent Non-reacting Propane Jet”, AIAA Journal, Vol.39, no.1, pp.64-72, 2001

Measuring Instrument	Quantities measured
Laser Doppler Anemometer (LDA)	Velocity & turbulence parameters
Rayleigh Scattering	Mixture fraction

Test case description (**Experimental Setup**)

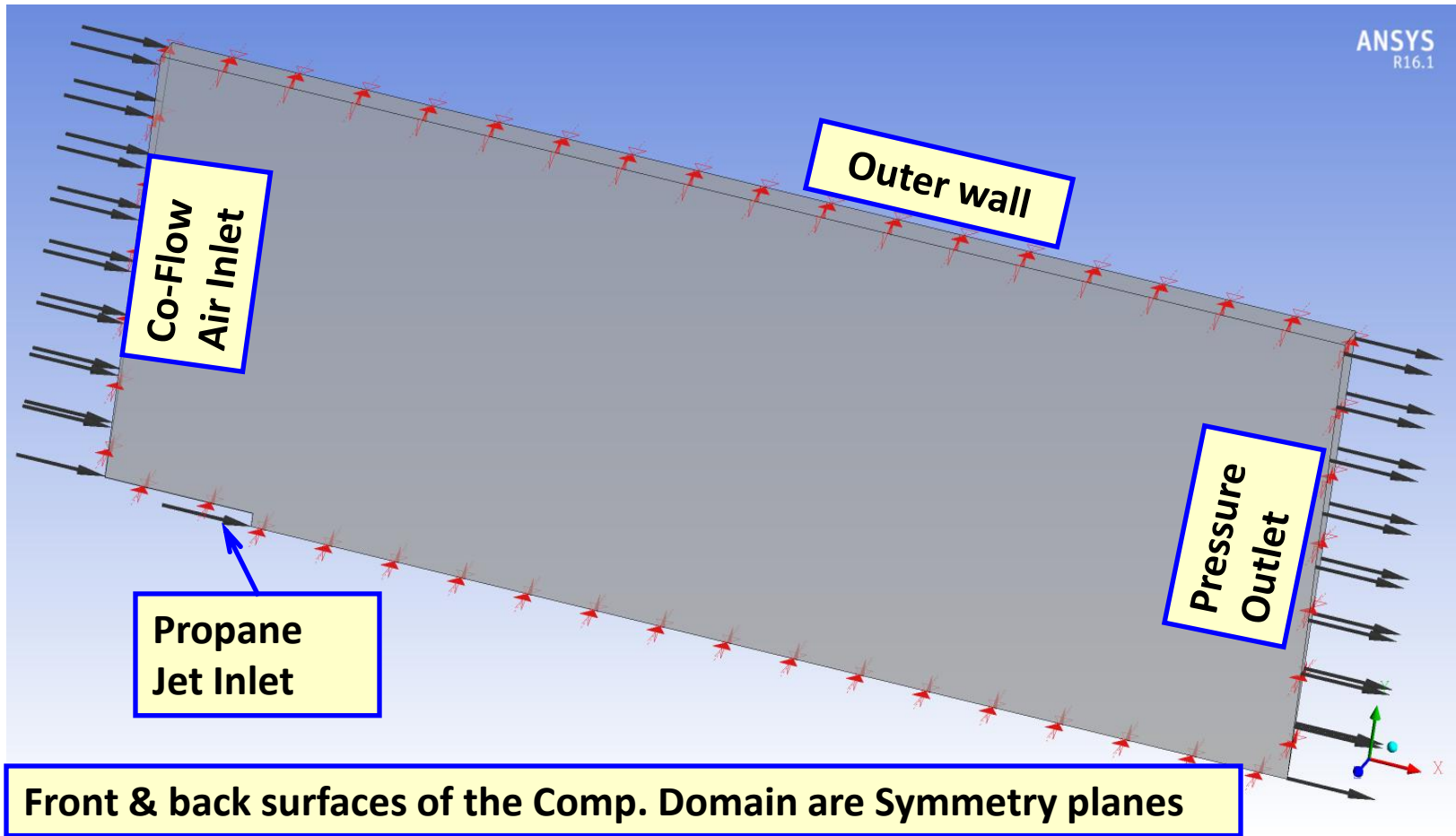


AXISYMMETRIC FUEL JET

Dimension of the test setup

Orientation	Vertical
Test Section	30 cm x 30 cm
Jet Tube Exit	0.52 cm (I. D.) 0.90 cm (O. D.)
Length of Fuel Jet Tube Straight Section Prior to Exit	2 m
Propane Jet Velocity	53 m/s (± 0.1 m/s)
Propane Jet Temperature	294 K (± 2 K)
Coflow Air Velocity	9.2 m/s (± 0.1 m/s)
Coflow Air Temperature	294 K (± 2 K)
Reynolds Number (based on jet exit diameter)	68,000
Coflow Air Turbulence	0.4%
Axial Pressure Gradient	6 Pa/m

Computational Domain



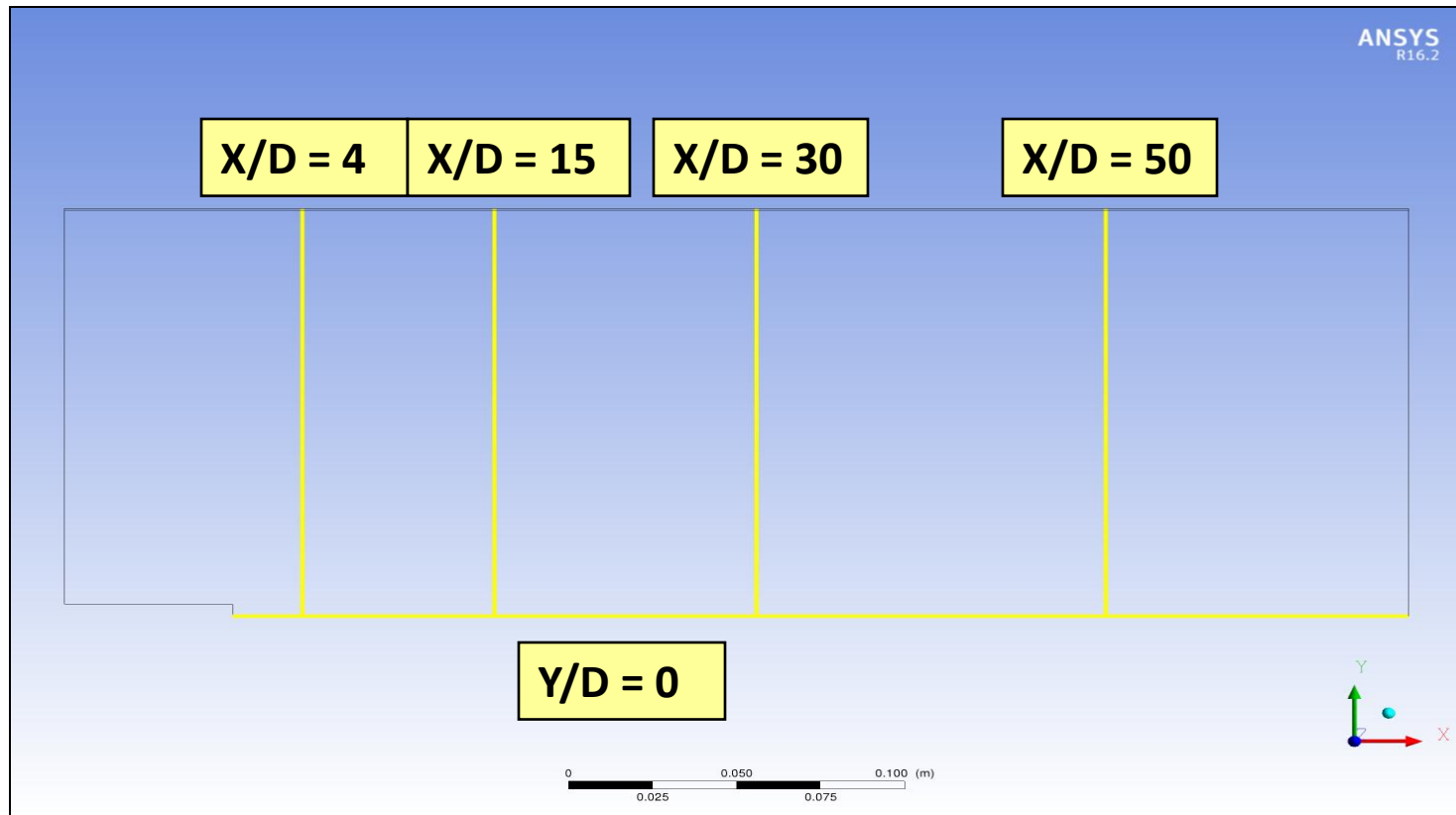
Computational Mesh

Parameter	Mesh 1	Mesh 2	Mesh 3	Mesh 4
Type of Mesh	Hexahedral	Hexahedral	Hexahedral	Hexahedral
Number of Nodes	3736	15568	63532	256660
Number of Elements	7665	31530	127860	514920
Max Aspect Ratio	22.57	24.54	25.59	26.52
Min Angle	90°	90°	90°	90°



Measured Quantities and Measurement locations

- Mean radial and axial velocity component (U , V)
- Mean mixture fraction



Boundary Condition

Surface Name	Boundary condition for ANSYS CFX 16.1	Boundary condition for ANSYS Fluent 17.0
Co-flow inlet	Normal Velocity = 9.2 [m/s] Turbulent Intensity = 0.5% Turbulent viscosity ratio = 10	Normal Velocity = 9.2 [m/s] Turbulent Intensity = 0.5% Turbulent viscosity ratio = 10
Fuel jet inlet	u, v, w, k, ε and ω interpolated from the imported fully developed pipe flow profile	u, v, w, k, ε and ω interpolated from the imported fully developed pipe flow profile
Outlet	Average static pressure Relative pressure = 0 [Pa] Pressure profile blend = 0.05	Average static pressure Gauge pressure = 0 [Pa] Pressure profile blend = 0.05
Bluff body	No slip wall	No slip wall
Outer wall	Free slip wall	Free slip wall
Symmetry 1 & 2	Symmetry	Symmetry

Quality Assurance

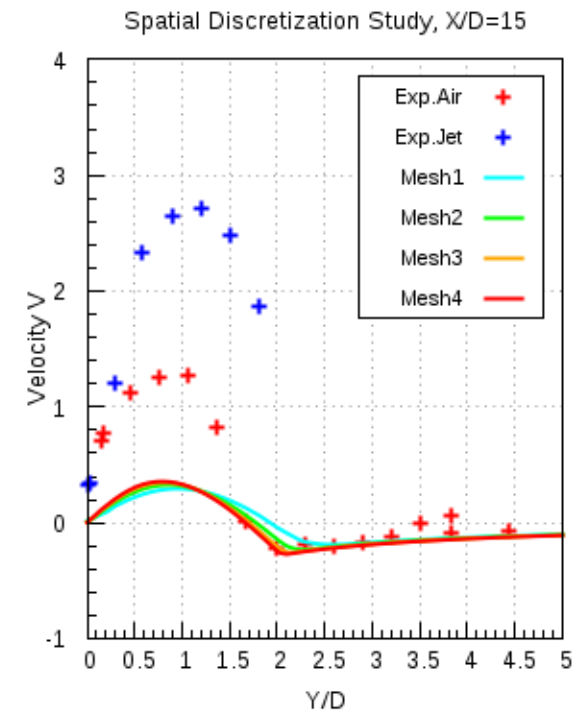
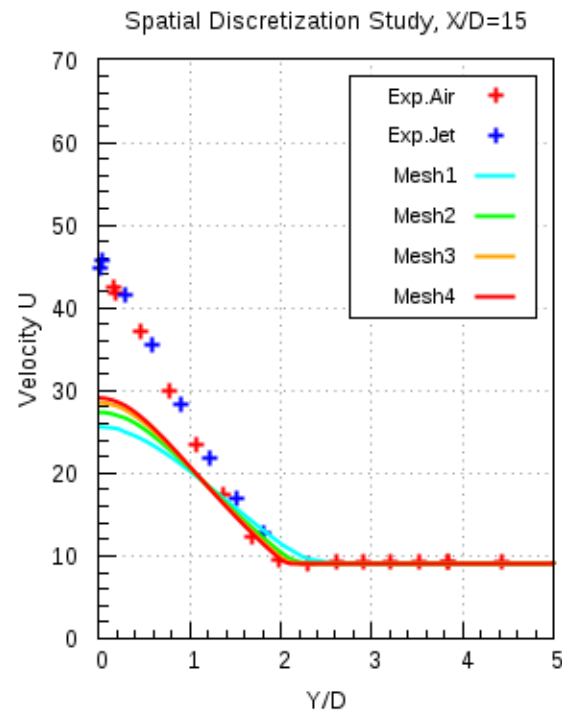
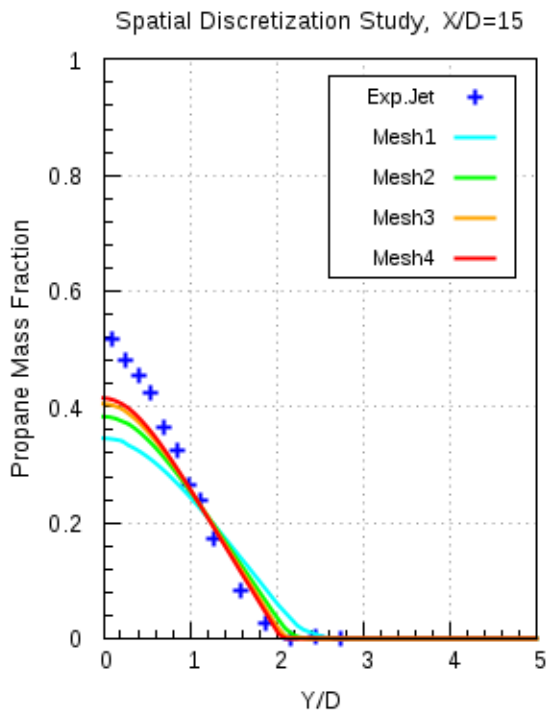
- The following investigations have been carried out in order to assure the quality of the CFD simulation:
 - Iteration error
 - dependence of CFD solution on the strength of the applied convergence criterion
 - Spatial Discretization error
 - grid independence of the CFD solution
 - Turbulent inlet boundary condition study
 - Turbulent Schmidt number (Sc_t) study
 - different Sc_t ranging from 0.6 – 0.9 are used to study with which Sc_t the result with best accuracy is delivered
 - Model error
 - comparison of SST, $k-\omega$, $k-\epsilon$, BSL RSM and EARSM turbulence models

Quality Assurance

- The following investigations have been carried out in order to assure the quality of the CFD simulation:
 - Iteration error
 - dependence of CFD solution on the strength of the applied convergence criterion
 - Spatial Discretization error
 - grid independence of the CFD solution
 - Turbulent inlet boundary condition study
 - Turbulent Schmidt number (Sc_t) study
 - different Sc_t ranging from 0.6 – 0.9 are used to study with which Sc_t the result with best accuracy is delivered
 - Model error
 - comparison of SST, $k-\omega$, $k-\epsilon$, BSL RSM and EARSM turbulence models

Spatial Discretisation Study (Mesh independency)

ANSYS CFX 16.1,
K- ω SST



Mesh 4 is chosen for further investigations

Test of turbulent Inlet Boundary Conditions (TIBC), Mesh 4

- Test to check the appropriate TIBC
- Turbulent kinetic energy (k) and length scale (LS) used
- Round jet anomaly phenomenon
- $K-\omega$ SST and Standard $K-\epsilon$ models are used in combination with different TIBC`s
- Propane mass fraction , velocity profiles compared with each other as well as with the experimental data

Test of Turbulence Inlet Boundary Condition (TIBC)

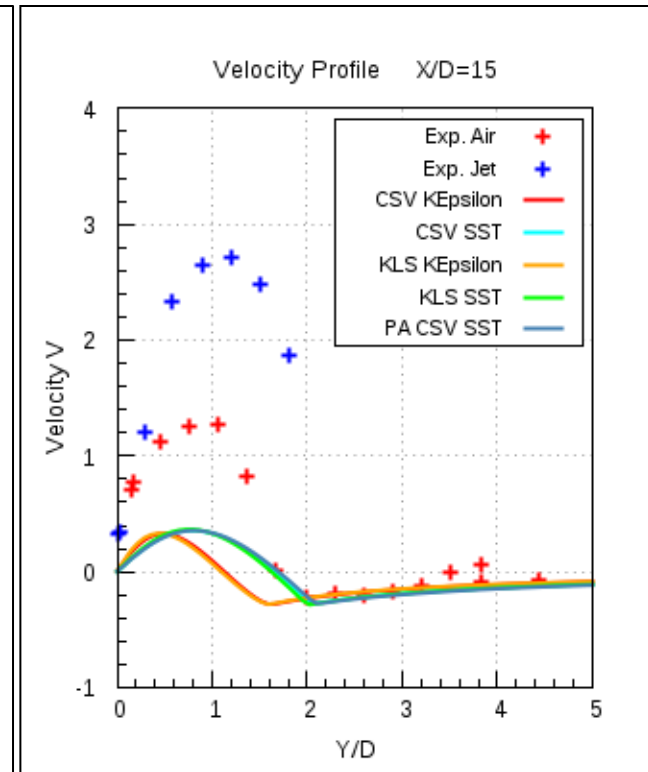
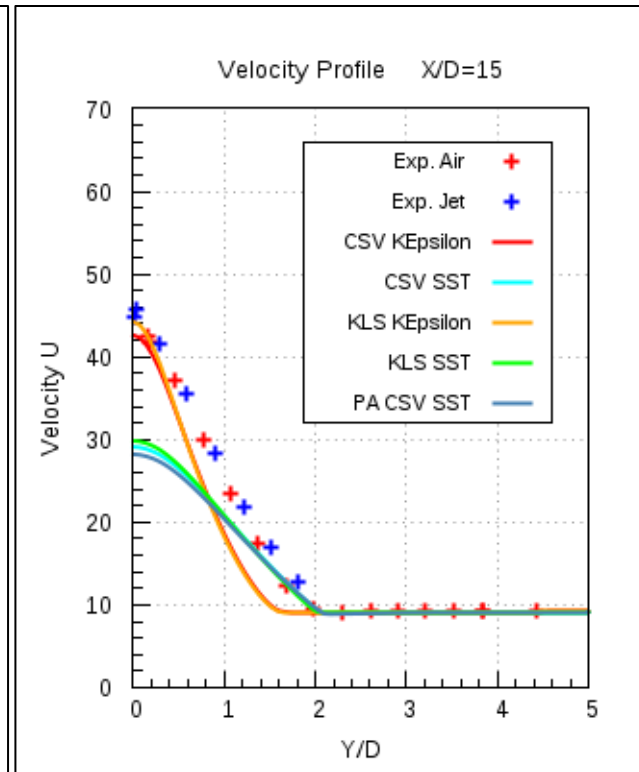
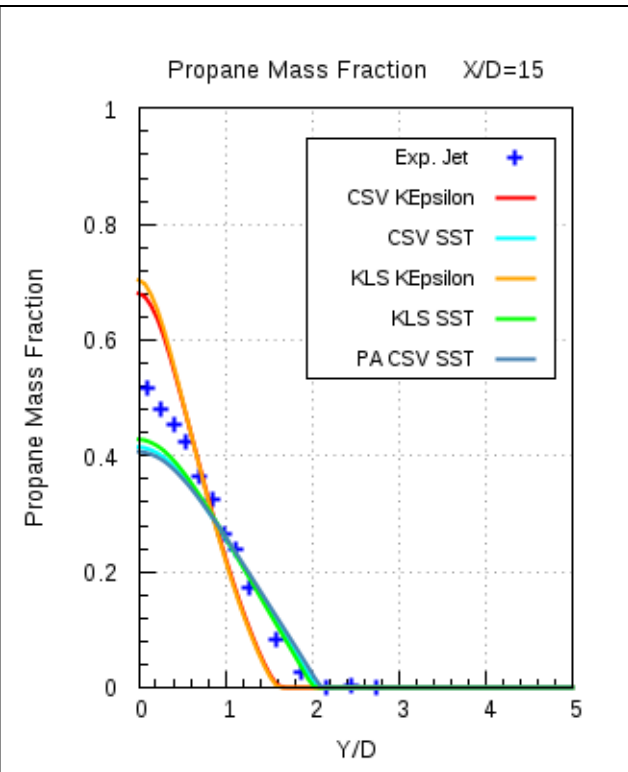
Test Name	Turbulence Model	Turbulence Boundary Condition Fuel Inlet	Turbulence Boundary Condition Air Inlet
CSV_SST	SST	K and ω	Fractional Intensity & Eddy Viscosity ratio
CSV_KEpsilon [#]	K- ϵ	K and ϵ	Fractional Intensity & Eddy Viscosity ratio
KLS_SST	SST	K and LS	K and LS
KLS_KEpsilon [#]	K- ϵ	K and LS	K and LS
Propane_Air_Inlet_ CSV**(PA CSV SST)	SST	K and ω	Fractional Intensity* & Eddy Viscosity ratio

Round Jet Anomaly: The standard k- ϵ model with the standard coefficients delivers the velocity field quite accurately in two-dimensional plane jet, but large errors occur for the axisymmetric jets. Specifically, the spreading rate of round jet is overestimated by 40% [Ref. Pope].

Reason: Modeling of dissipation (ϵ) equation

TIBC – Results comparison

ANSYS CFX 16.1



TIBC – Results comparison

- For the purpose of comparison, two new non-dimensionalised quantities were introduced to compare the velocity and the mass fraction decay along the axial direction.
 - Velocity u invariant (U invar)**

$$\left(\frac{U_{cl} - U_{air}}{\text{Velocity } u - U_{air}} \right)$$

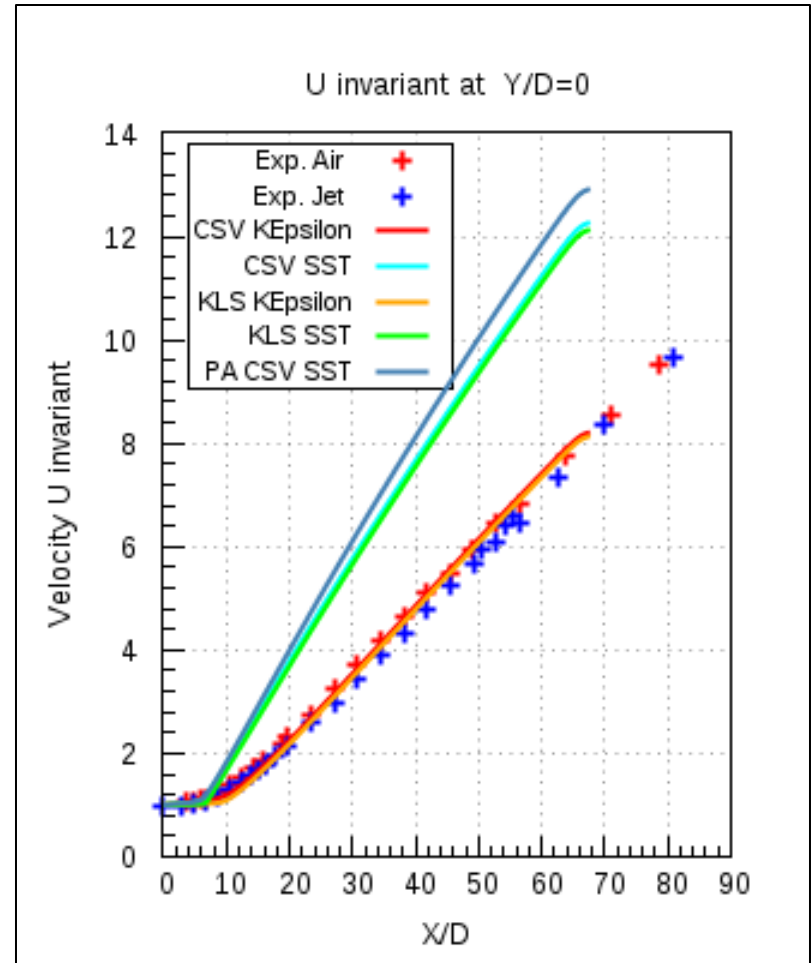
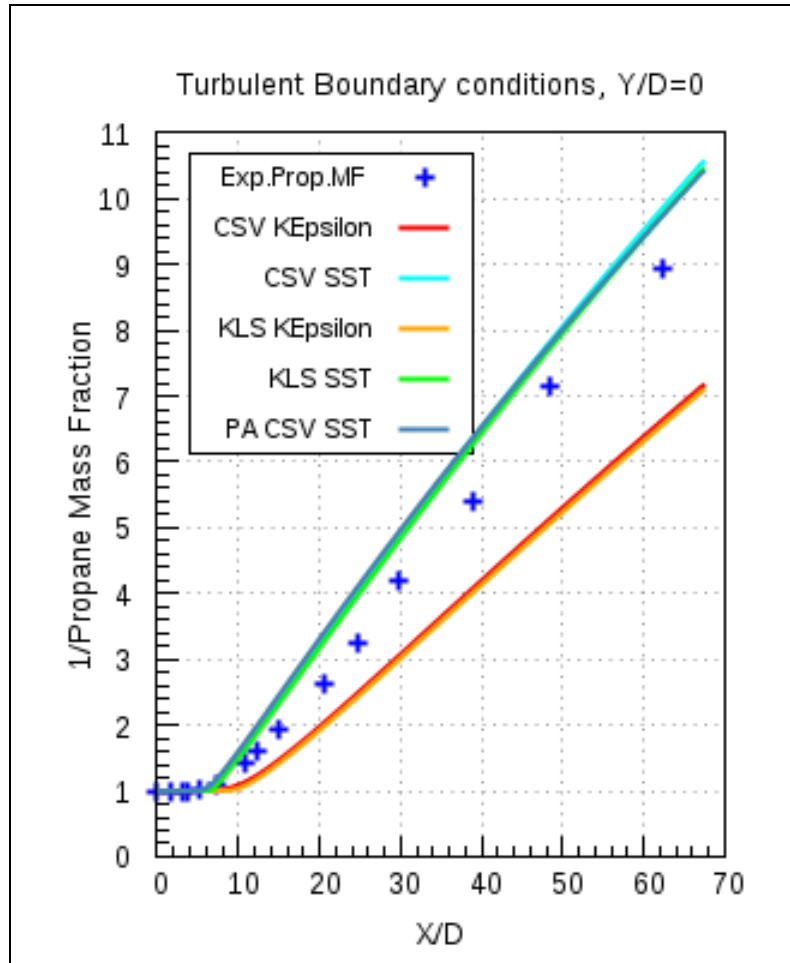
where, U_{cl} is the centerline velocity, U_{air} is the constant velocity of co-flow air 9.2 m/s, **Velocity u** is the result obtained from the numerical calculation.

- Propane mass fraction invariant (PMF invar)**

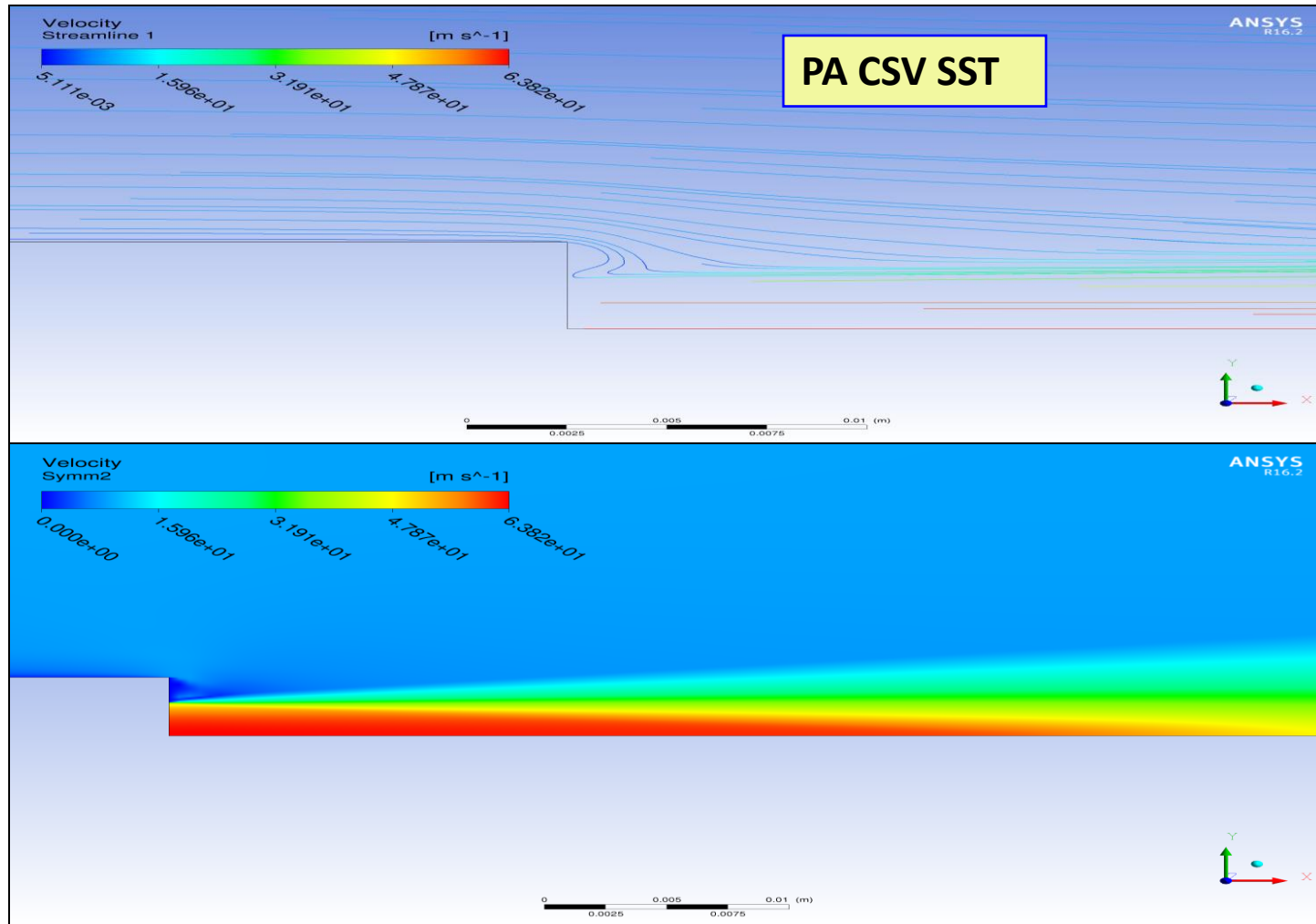
$$\left(\frac{1}{\text{Propane mass fraction}} \right)$$

TIBC – Results comparison

ANSYS CFX 16.1



TIBC – Results



Model Error Study, Mesh 4

For this study the following turbulent models are used :

- K- ω SST
- K- ϵ
- K- ϵ modified
- BSL RSM (Base line reynolds stress model)
- EARSM (Explicit algebraic reynolds number)

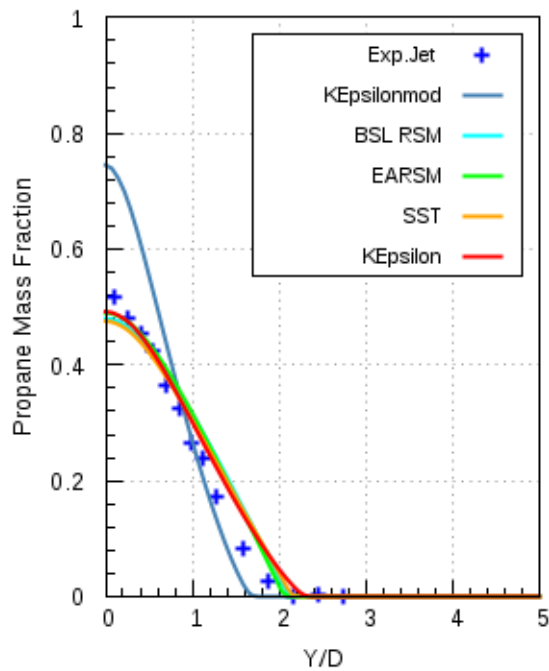
Resulting profiles for the propane mass fraction and velocities are compared with the other turbulence models and the experimental data

Model Error - Results

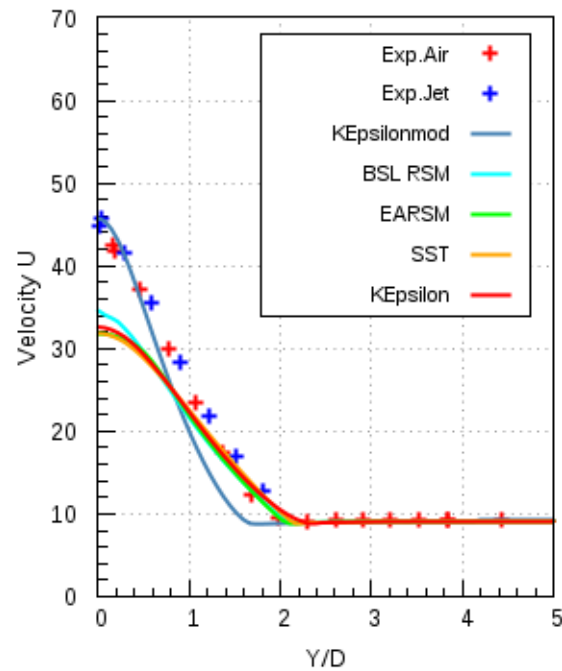
ANSYS CFX 16.1



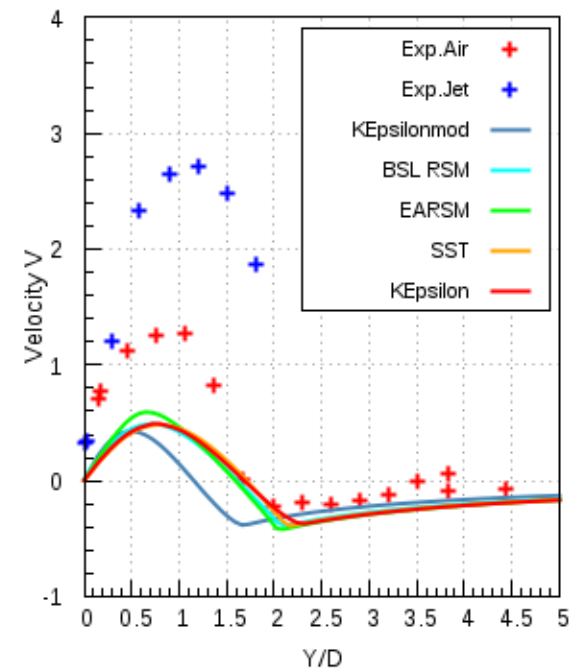
Model Error Study $X/D=15$



Model Error Study, $X/D=15$

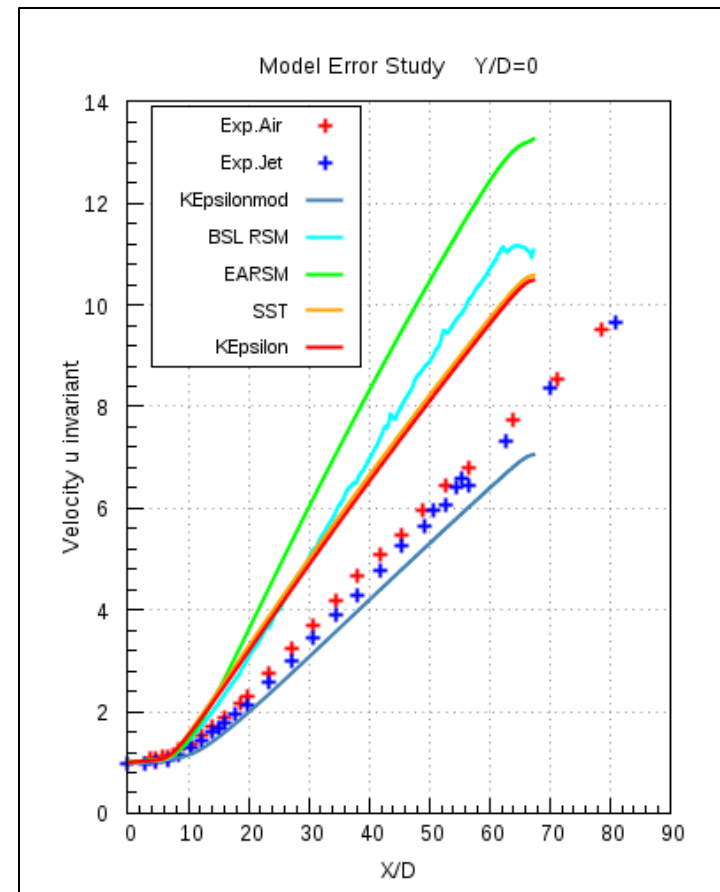
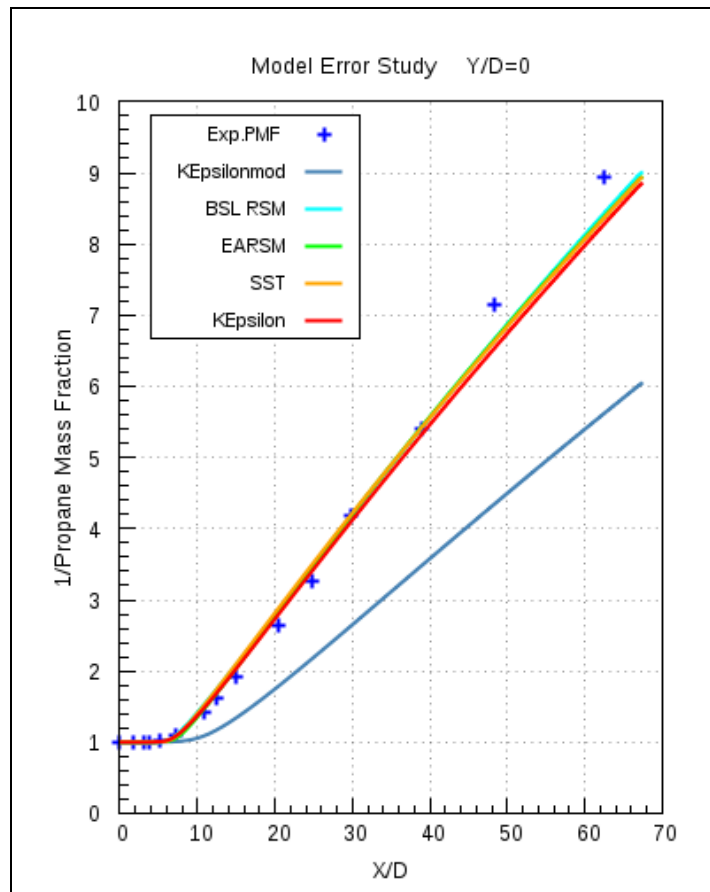


Model Error Study, $X/D=15$



Model Error- Results

ANSYS CFX 16.1

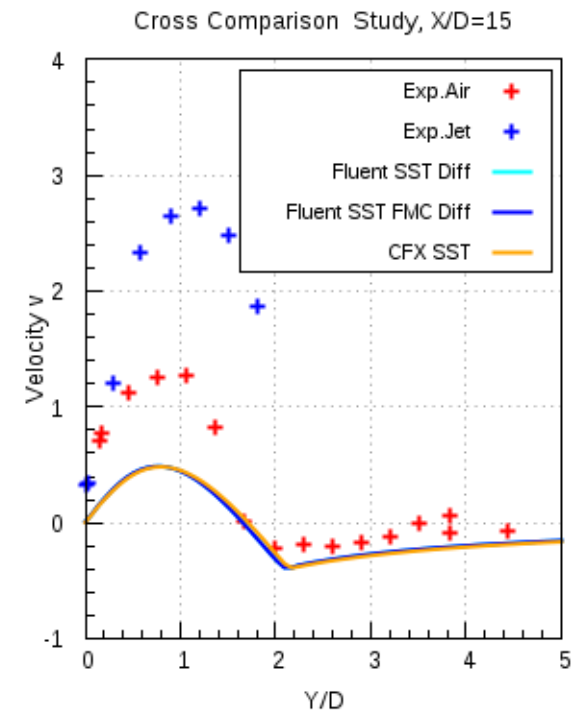
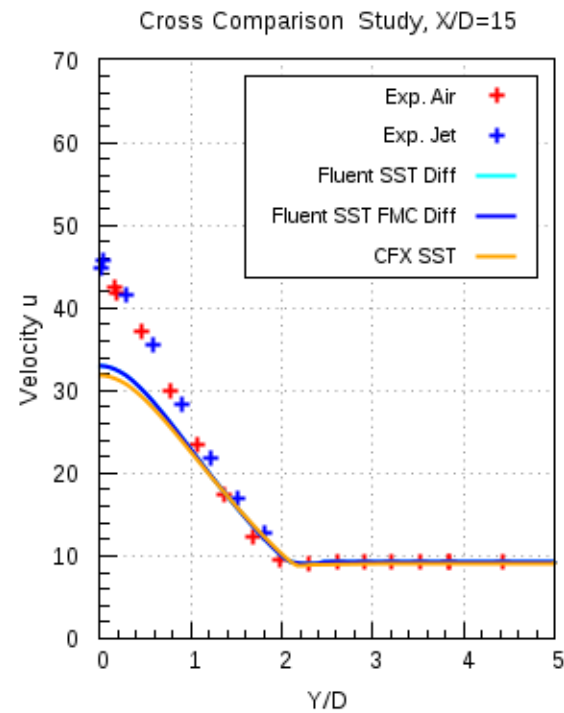
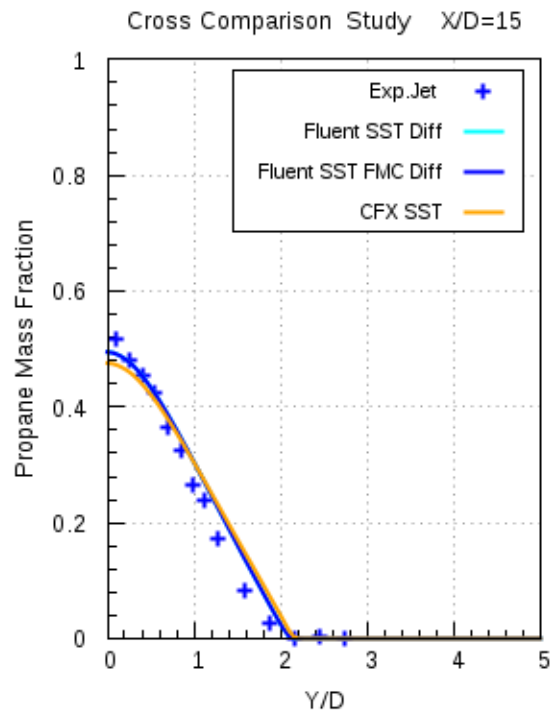


Solver Comparison between ANSYS CFX 16.1 and ANSYS Fluent 17.0, Mesh 4

- The following two turbulence models are used for the solver comparison
- $K-\omega$ SST
- $K-\varepsilon$

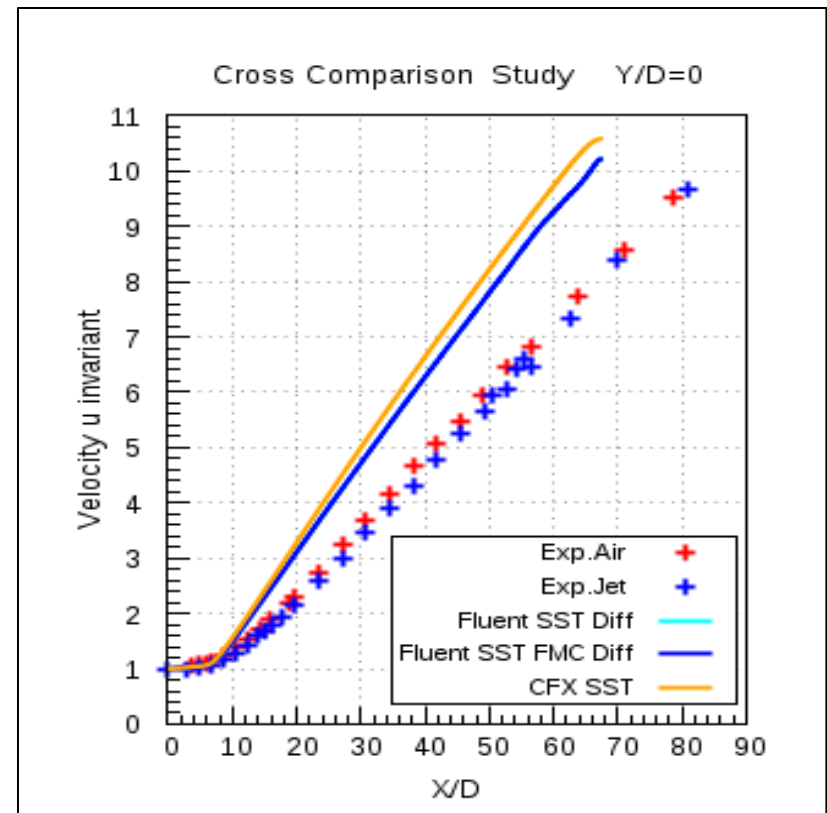
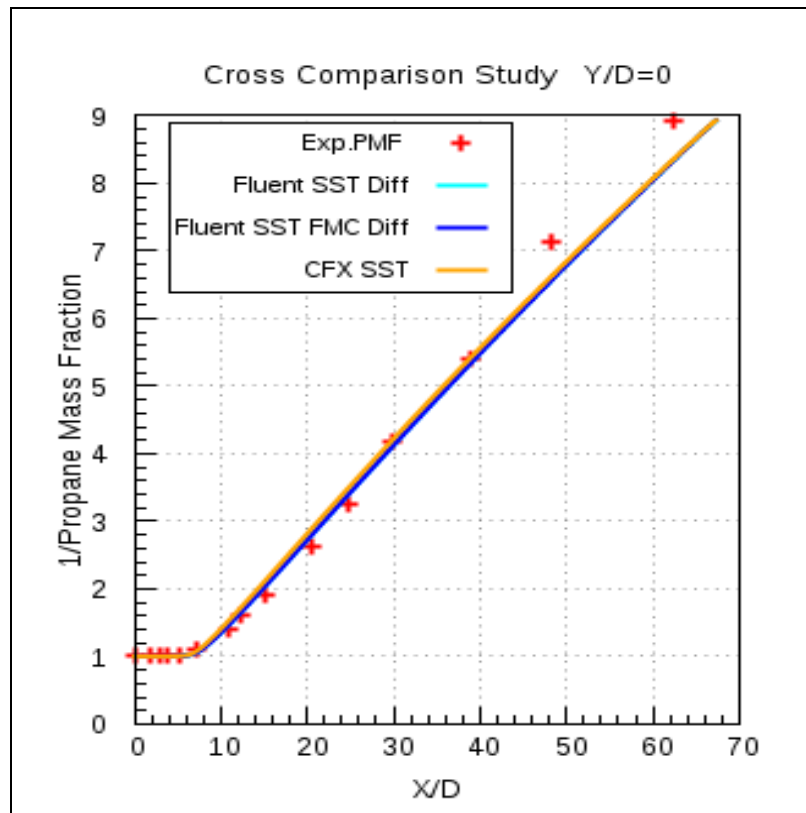
Solver Comparison

ANSYS CFX 16.1 and ANSYS Fluent 17.0 k- ω SST



Solver Comparison

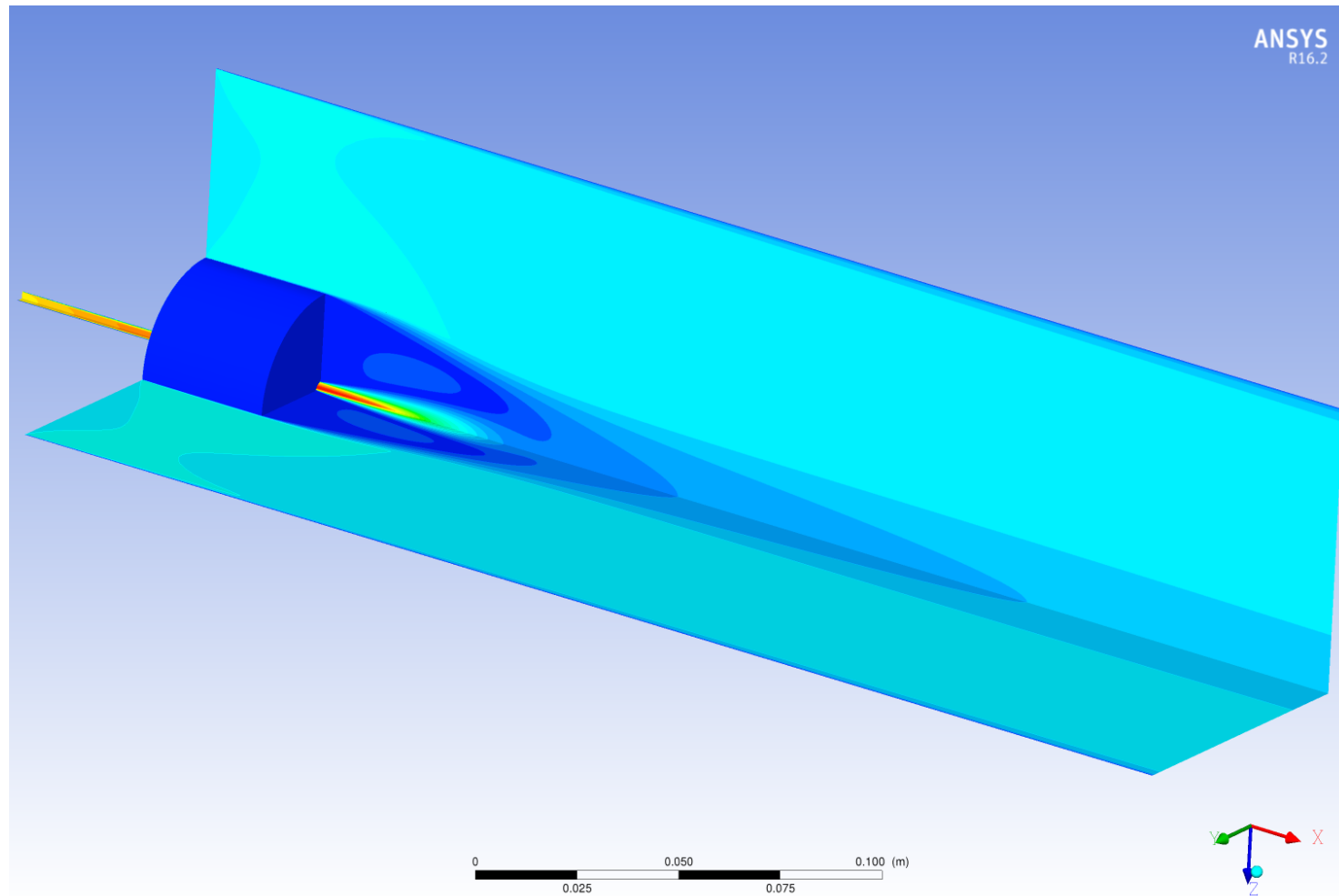
ANSYS CFX 16.1 and ANSYS Fluent 17.0, $k-\omega$ SST



Conclusion

- In this test case the investigation of the turbulent mixing of species in the recirculation zone behind the bluff body was carried out with **isothermal** and **steady state** flow conditions
- **K- ω SST** model would be the recommended turbulence model. Both **k- ϵ** models give almost the same level of accuracy in this investigation, but with more difficulties in obtaining a converged solution
- Differences observed in the axial velocity profiles is due to the **simplifying assumptions** made in the RANS turbulence models and also due to the **round jet anomaly**
- Modification in the model constants is not suggested
 - **improves** comparison of velocity profiles
 - but at the price of **deteriorated** Propane mass fraction profiles
- Turbulent Schmidt number (Sc_t) for the transport equation of species should be **0.9**
- If desired, **higher accuracy** results can be obtained with e.g. Large Eddy Simulation (LES) or hybrid models, but at much higher computational costs

Part II - Sydney Bluff Body Jet



Experimental References

- **Main Reference:**

- The University of Sydney, Department of Mechanical and Mechatronic Engineering and Sandia National Laboratories Combustion Research Facility, “Turbulent Non premixed Combustion Data for Piloted and Bluff-Body Stabilized Flows”
- <http://sydney.edu.au/engineering/aeromech/thermofluids/bluff.htm>

- **Secondary Reference:**

- Dally, Bassam B., Masri, Assaad R., Fletcher, David F. “Modeling of bluff body recirculating flows”, Twelfth Australasian Fluid Mechanics Conference, The University of Sydney, Australia, pp. 529-532, 1995
- B.B.Dally, D F Fletcher & A.R.Masri, “Flow and mixing fields of turbulent bluff-body jets and flames”, Combustion Theory and Modeling 2, pp.193-219, 1998

Experimental Setup

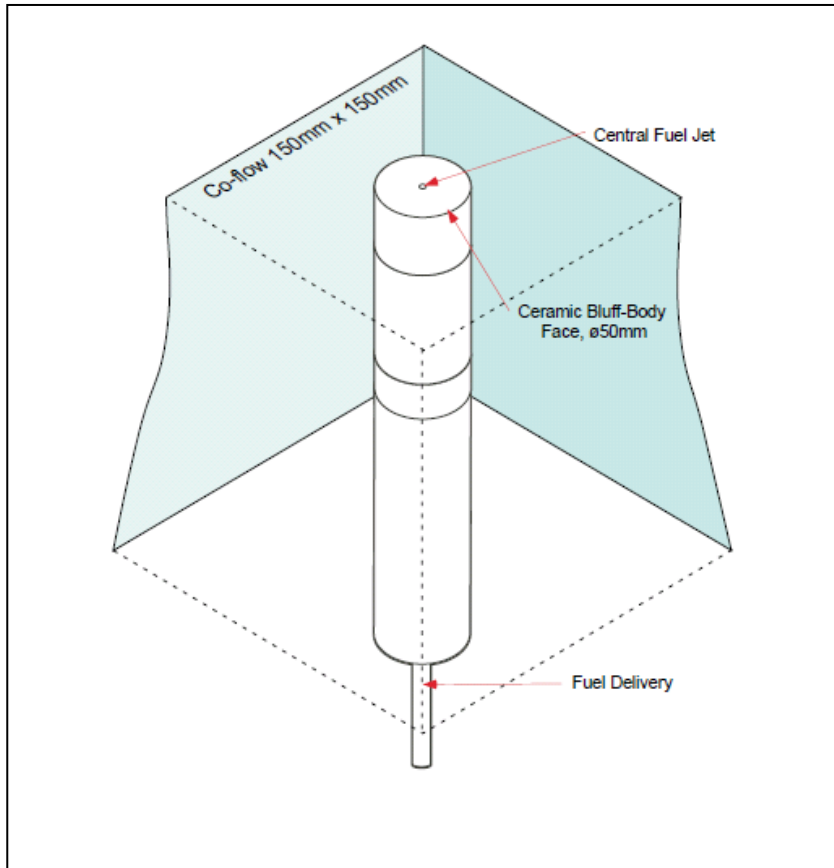


Fig (a) Sydney bluff body burner placed in the wind tunnel

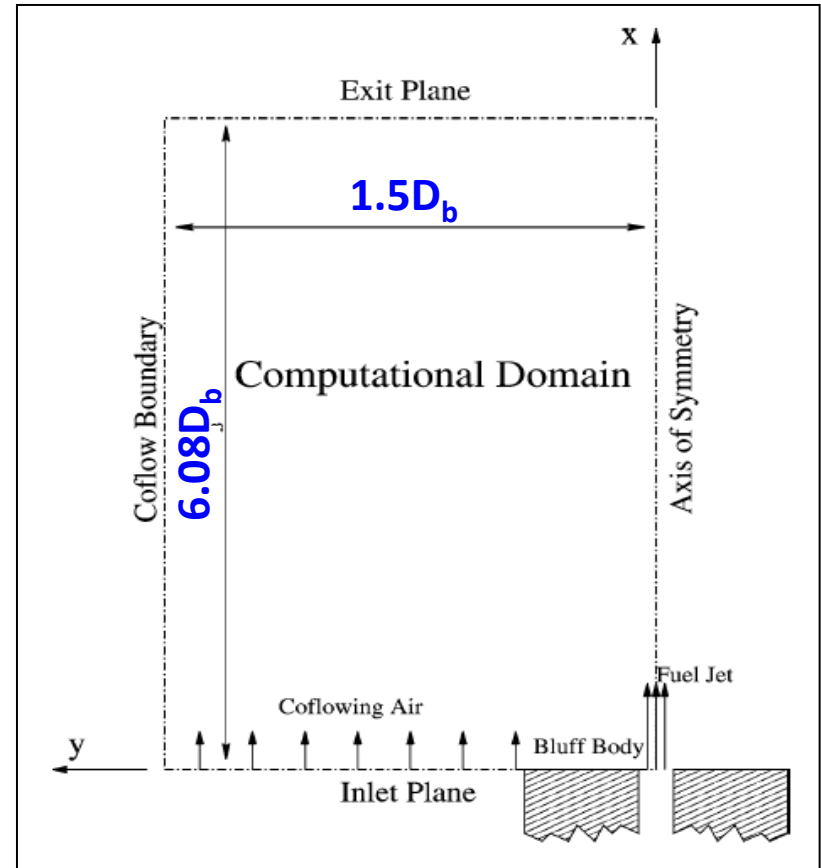


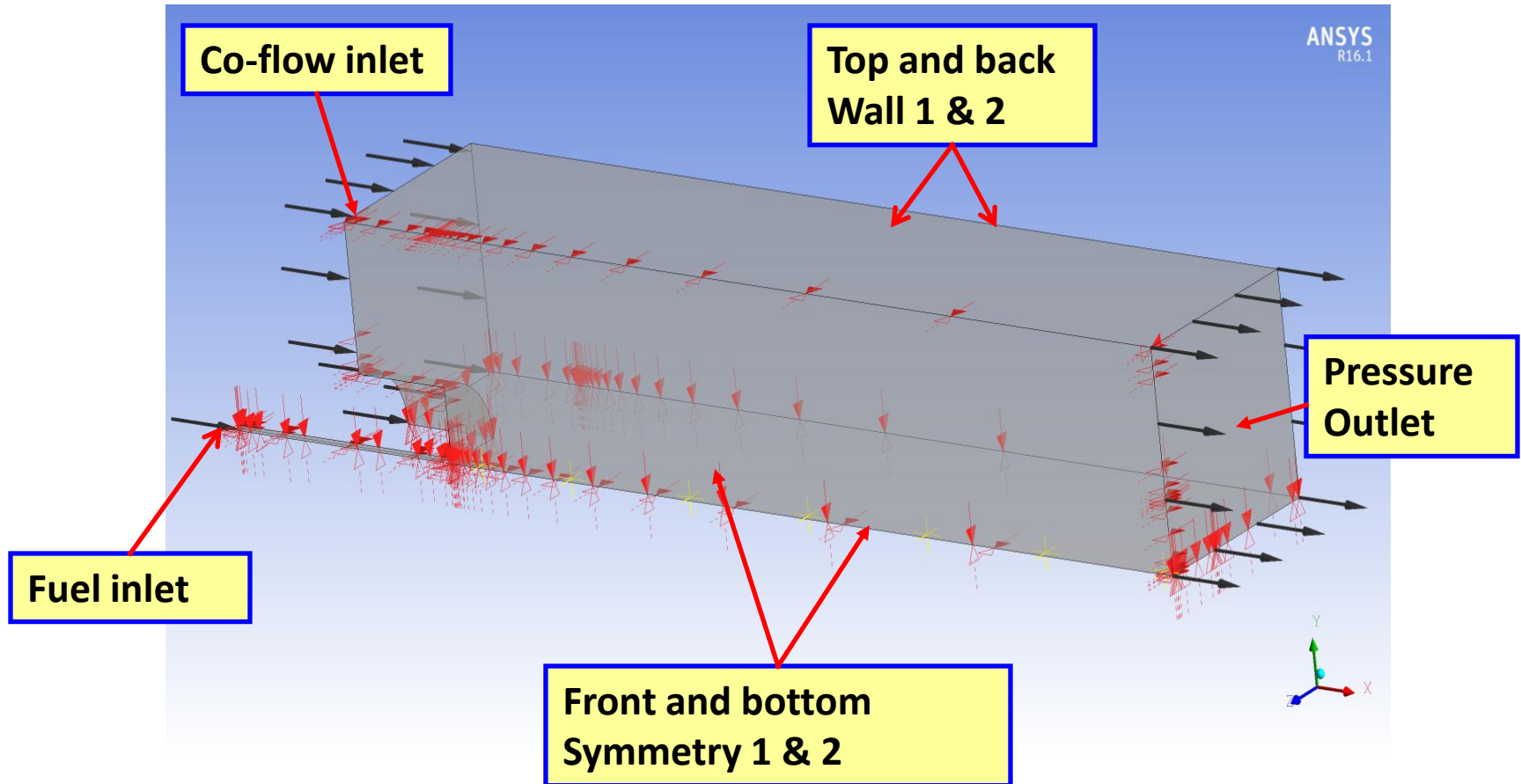
Fig (b) 2D axis-symmetric experimental setup

Experimental data

- Experiment has been performed in two parts
 - **Constant density flow** : Velocity profiles were measured
 - **Variable density flow** : Mixture fraction profiles were measured
- Air velocity in co-flow is $20 \text{ [m s}^{-1}\text{]}$ in all cases
- For fuel different velocities are being considered for different fuels:
 - Air : $61 \text{ [m s}^{-1}\text{]}$
 - CNG : $50, 85, 143 \text{ [m s}^{-1}\text{]}$
 - Ethylene : $50, 63, 80 \text{ [m s}^{-1}\text{]}$
 - LPG : $50, 70 \text{ [m s}^{-1}\text{]}$
- Temperature T_{inlet} : 298 [K]

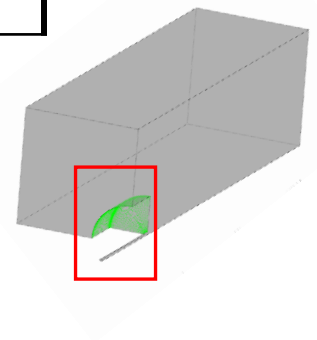
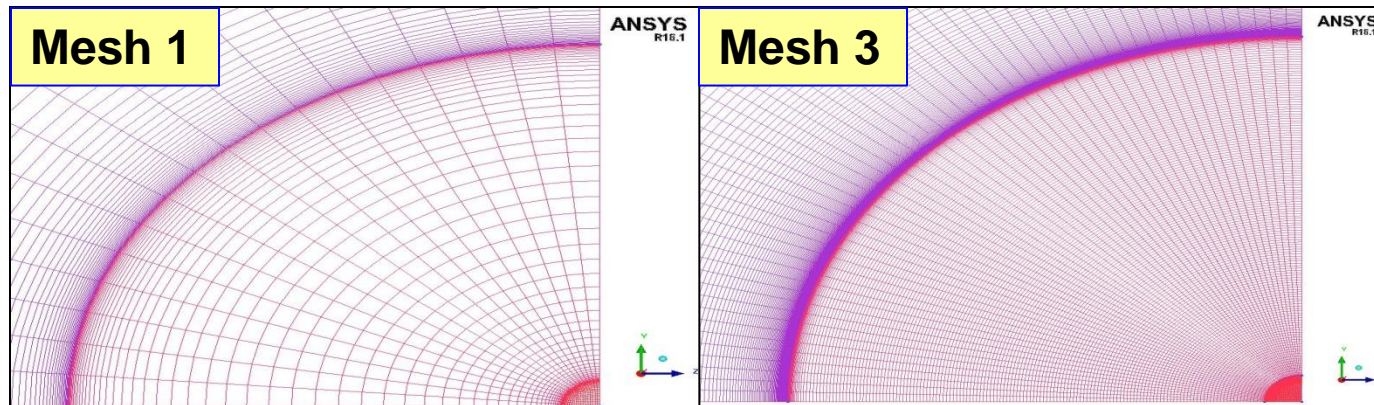
Measurement Technique	Measured physical quantity
Laser Doppler Velocimetry (LDV)	Velocities $\text{[m s}^{-1}\text{]}$
Planar imaging of Rayleigh scattering	Mixture fraction

Computational Domain



Computational Domain & Meshes

	Mesh 1	Mesh 2	Mesh 3
Elements	445779	3649862	12367909
Nodes	477248	3773055	12648672
Max. Aspect Ratio	665	1864	3046
Min. Grid Angle	45°	44.19°	40.95°



Boundary Conditions (**Variable density**)

Name	ANSYS CFX 16.1	ANSYS Fluent 17.0
Co-flow inlet	Normal Velocity = 20 [m/s] Turbulent Intensity = 2 % Turbulent viscosity ratio = 10	Normal velocity = 20 [m/s] Turbulent Intensity = 2 % Turbulent viscosity ratio = 10
Fuel Inlet	Normal Velocity = 50* [m/s] Turbulent Intensity = 5 % Turbulent viscosity ratio = 10	Normal Velocity = 50* [m/s] Turbulent Intensity = 5 % Turbulent viscosity ratio = 10
Outlet	Average static pressure Relative pressure = 0 [Pa] Pressure profile blend = 0.05	Average static pressure Gauge pressure = 0 [Pa] Pressure profile blend = 0.05
Bluff body	No Slip wall	No Slip wall
Wall 1 & 2	No Slip wall	No Slip wall
Symmetry 1 & 2	Symmetry	Symmetry

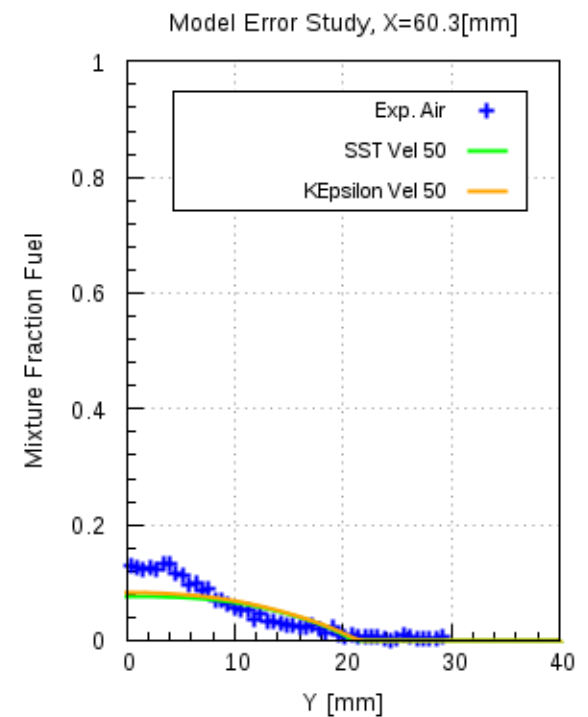
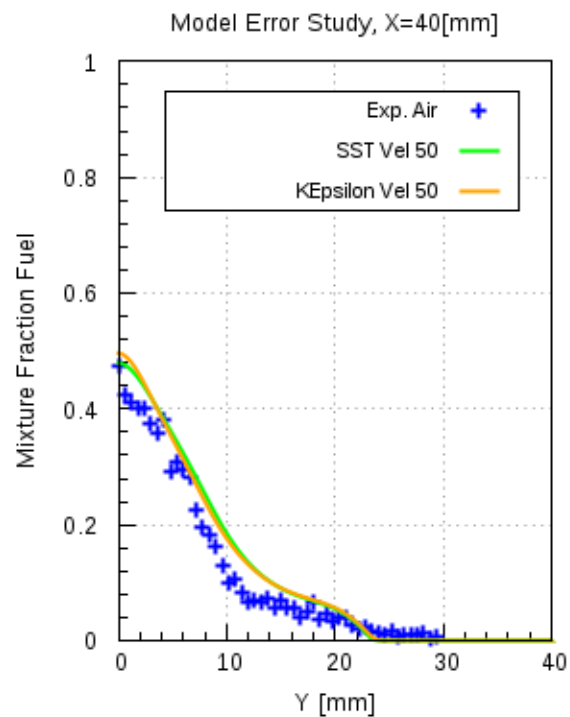
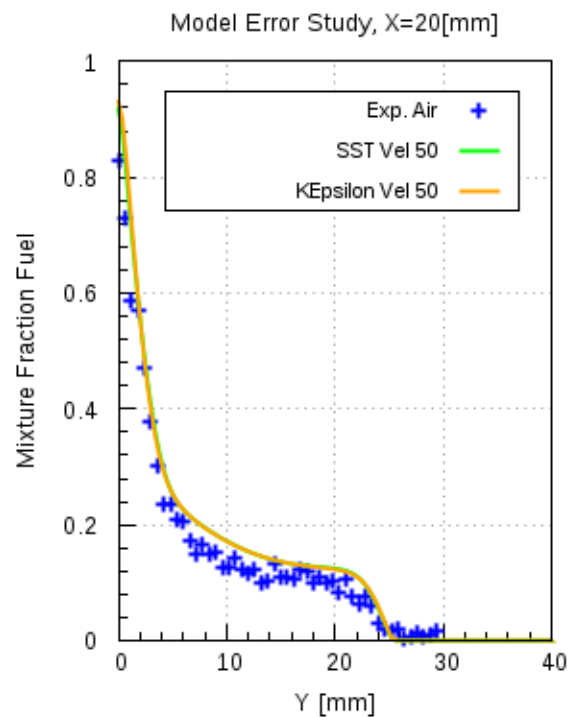
***Note:** The velocities at jet inlet will depend on the experiments performed as mentioned in the literature

ANSYS CFX 16.1 Model Error, Mesh 2

- Comparison of the mixture fraction profiles with the experimental data.
- The following turbulence models:
 - K- ω SST
 - k- ϵ
- The results showed are computed with fuel CNG

Model Error

CNG, Jet velocity = 50 m/s



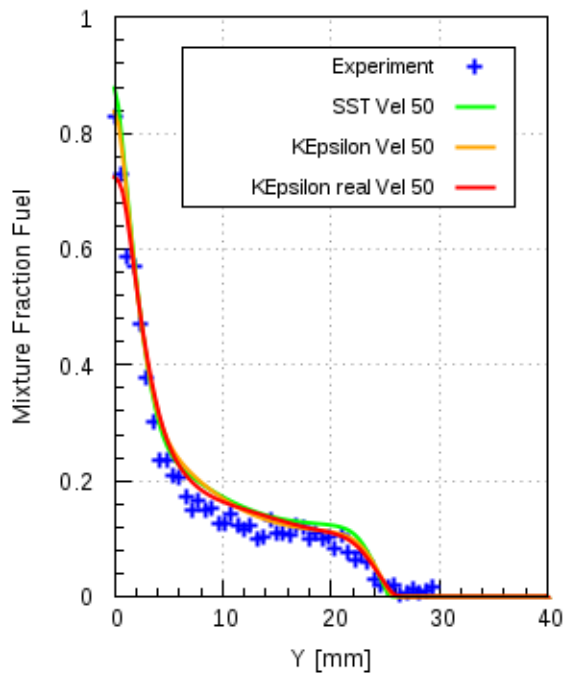
ANSYS Fluent 17.0 Model Error, Mesh 2

- Comparison of the mixture fraction profiles with the experimental data.
- The following turbulence models:
 - $k-\omega$ SST
 - standard $k-\varepsilon$
 - realizable $k-\varepsilon$
- The results showed are computed with fuel CNG

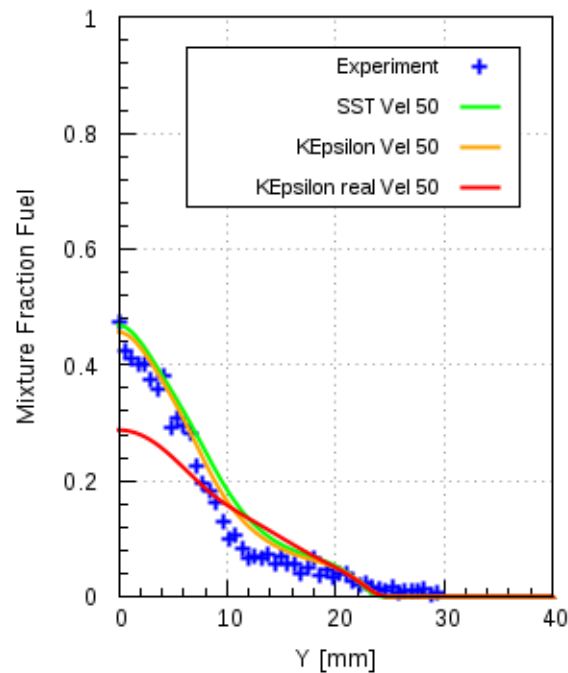
Model Error

CNG, Jet velocity = 50 m/s

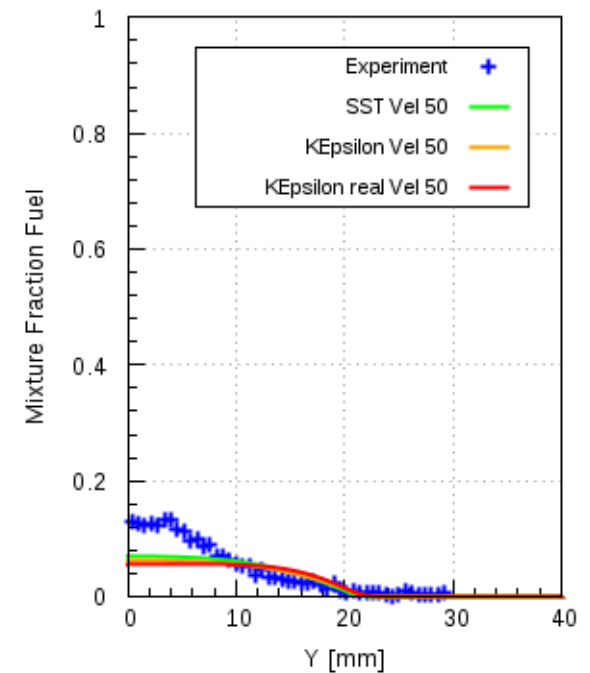
Radial Profile, X=20[mm]



Radial Profile, X=40[mm]



Radial Profile, X=60.3[mm]



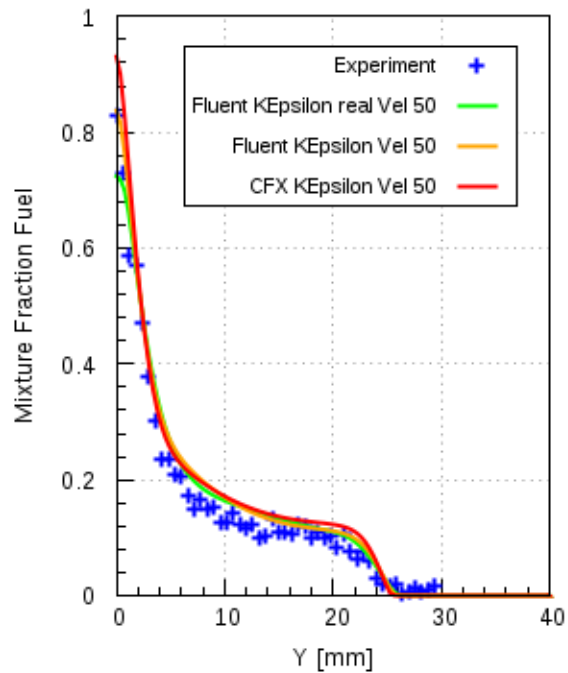
Solver Comparison between ANSYS CFX 16.1 and ANSYS Fluent 17.0, Mesh 2

- The following turbulence models are used for the solver comparison
 - $k-\omega$ SST
 - standard $k-\varepsilon$
 - realizable $k-\varepsilon$
- The results shown here are computed with fuel CNG with Jet velocity 50 m/s, with standard $k-\varepsilon$ and realizable $k-\varepsilon$ model

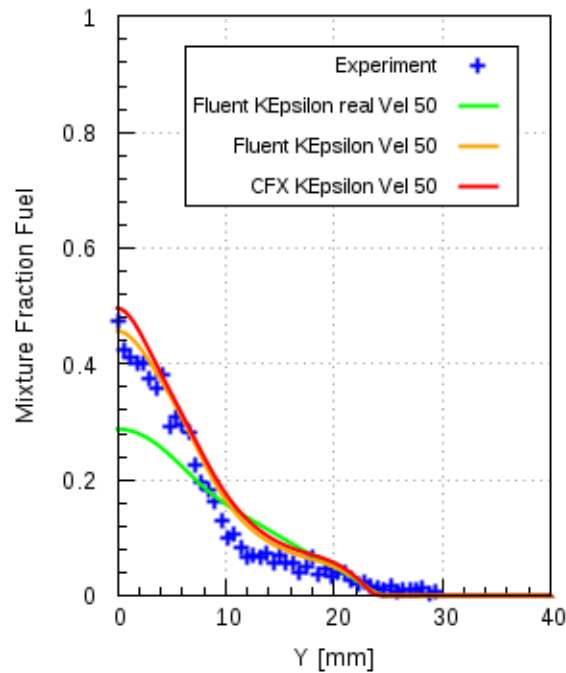
Solver Comparison

ANSYS CFX 16.1 and ANSYS Fluent 17.0, CNG

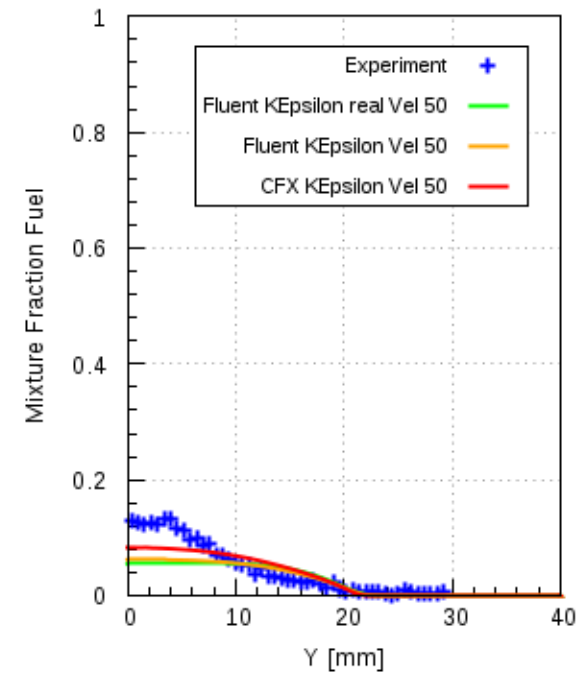
Radial Profile, X=20[mm]



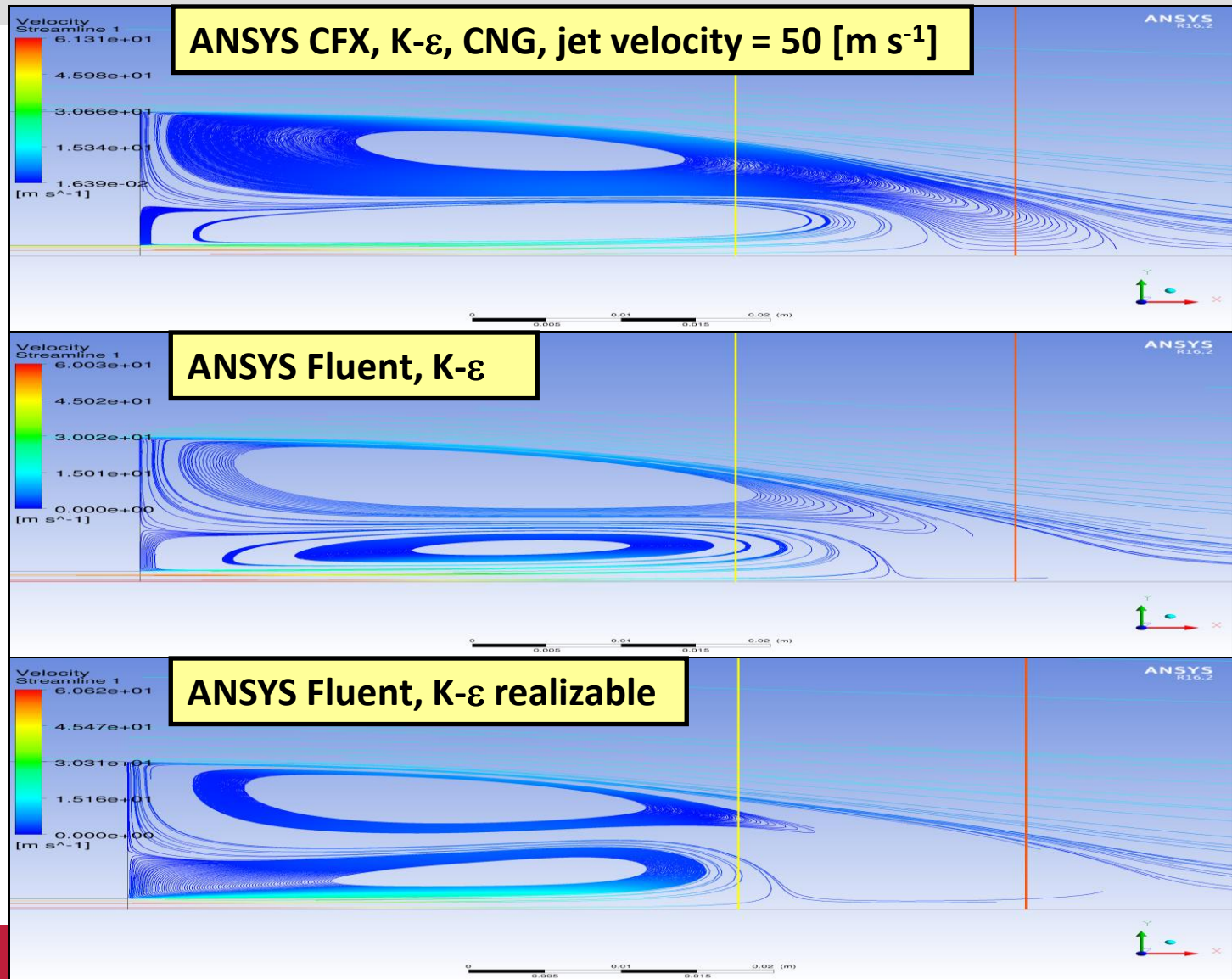
Radial Profile, X=40[mm]



Radial Profile, X=60.3[mm]

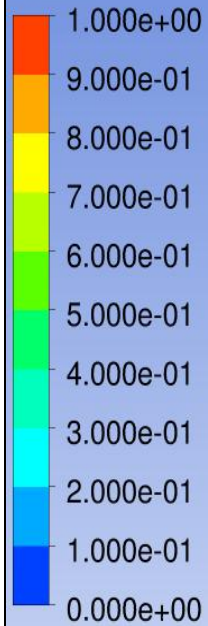


Solver Comparison

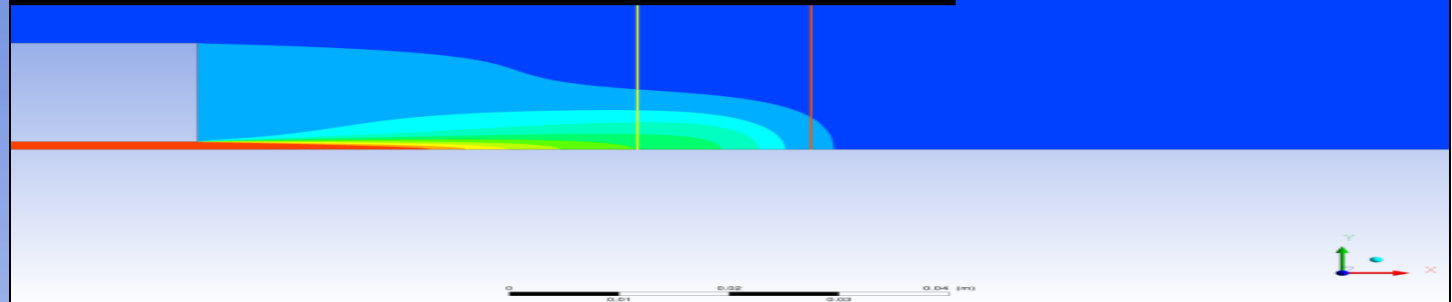


Solver Comparison

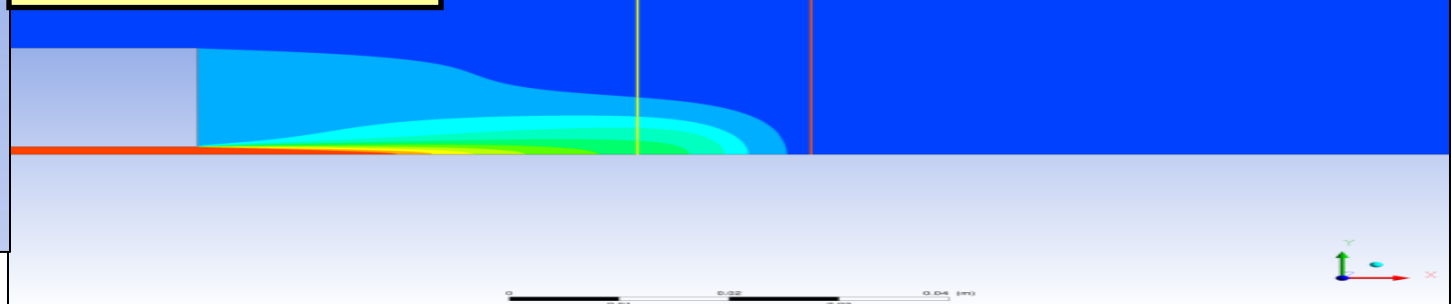
Mixture Fuel.Mass Fraction
Contour 1



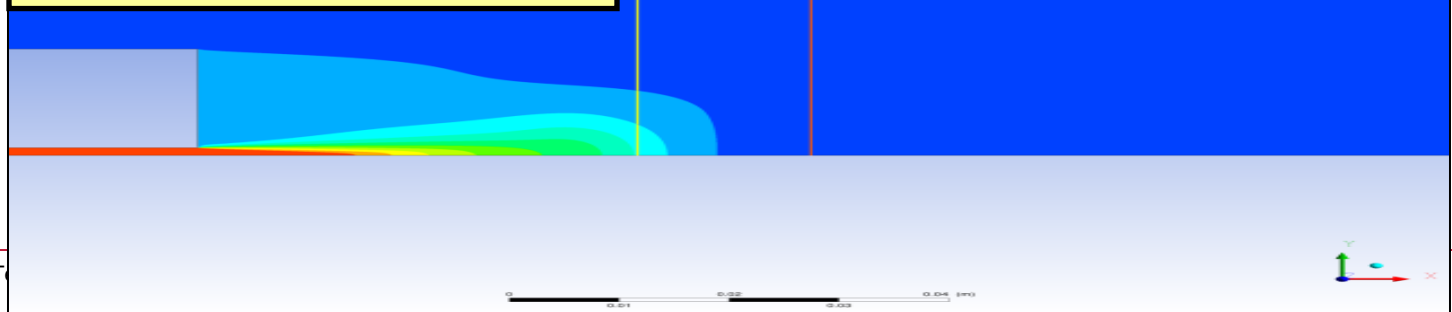
ANSYS CFX, K- ϵ , CNG, jet velocity = 50 [m s⁻¹]



ANSYS Fluent, K- ϵ



ANSYS Fluent, K- ϵ realizable



Conclusion

- Two fully developed vortices are present in the recirculation zone
- $k-\omega$ SST and standard $k-\epsilon$ model are the recommended turbulence models for the case of non-reacting turbulence mixing of the species
- Realizable $k-\epsilon$ model over-predicts the mixing at certain locations in the downstream direction, specifically in case of CNG it is more prominent
- Apart from that for all investigated cases the solutions of ANSYS CFX 16.1 and ANSYS Fluent 17.0 are in fairly good agreement with each other, as well as with the experimental data
- Also, a comparison of the size of the recirculation zone, behind the bluff body, obtained from numerical results and experiment showed that they are of similar size (~ 50 [mm] as mentioned in the secondary reference)

Thank you for your attention!

