Northern Illinois University

A Brief Tutorial of Computational Fluid Dynamics Using ANSYS Workbench

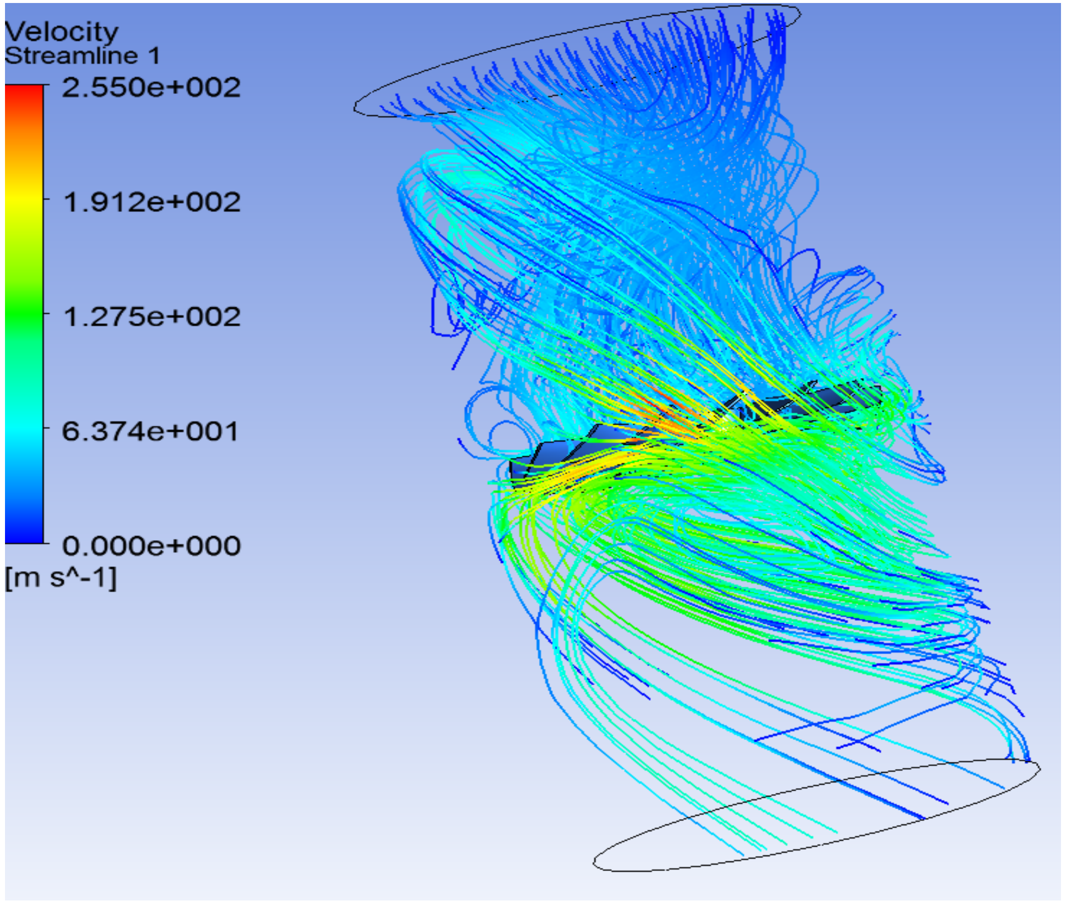
Group 1:
Sasanka Garapati
Adam Vann
Kunal Giridhar
Taylor Norton

Overview

- Introduction to CFD and tools in Ansys workbench
- Part 1- Problem Statement
- Geometry and Mesh details
- Boundary Conditions and solver inputs
- Results discussion
- Part 2 – Problem Statement
- Geometry and mesh details
- Boundary conditions and solver inputs
- Results discussion

Introduction to CFD

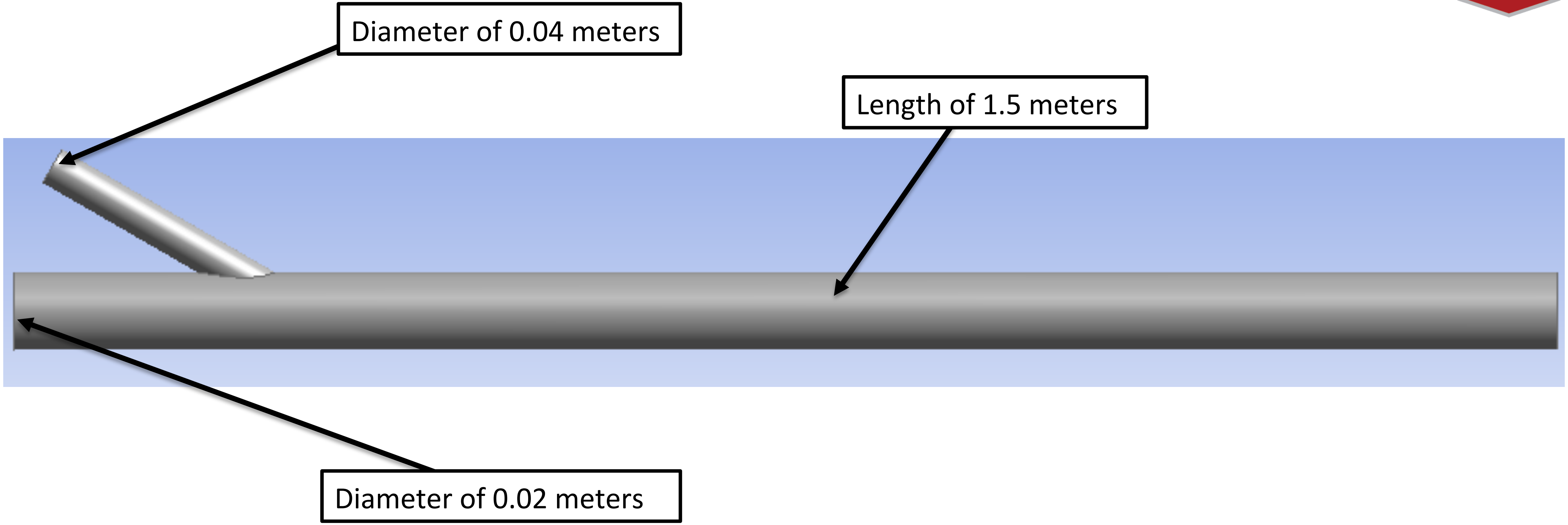
- CFD is the science of predicting fluid flow, heat transfer, mass transfer and chemical reactions.
- Governing equations of conservation of mass, momentum and energy are coupled.
- These are solved computationally to obtain approximate solution.



Tools in Ansys Workbench

- Two Independently developed solvers with many similarities.
- CFX – User friendly and easy to use.
- Fluent – Better approximation for multi-phase and radiation models.
- They differ mainly in the way they integrate the fluid flow equations and in their equation solution strategies.

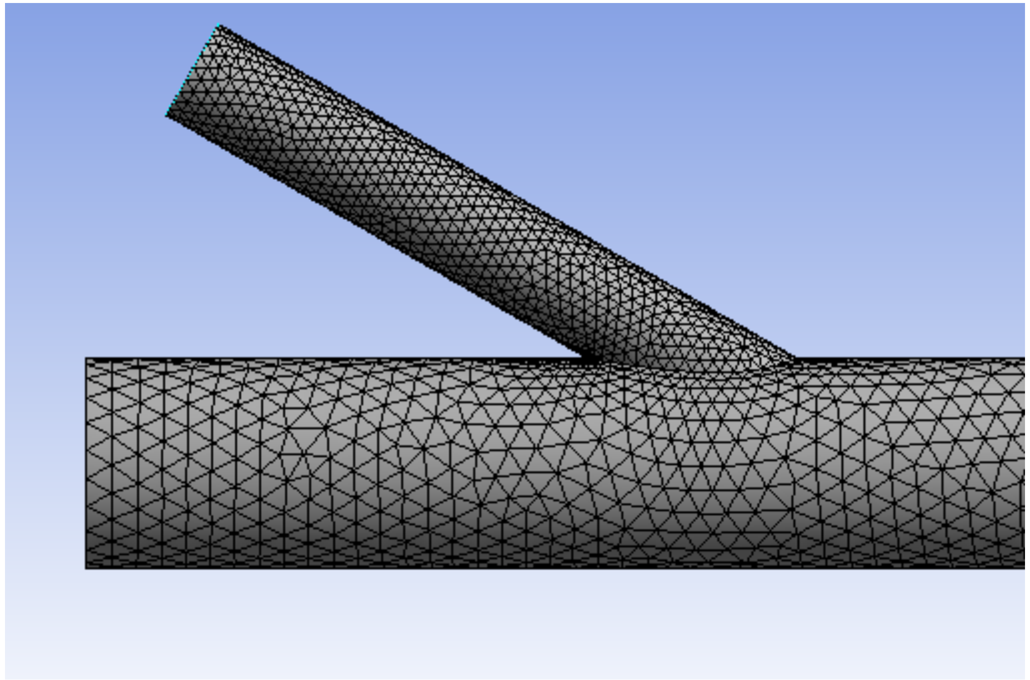
Mixing Pipe (Geometry)



Mixing Pipe (Mesh)

Physic Preference from mechanical to CDF

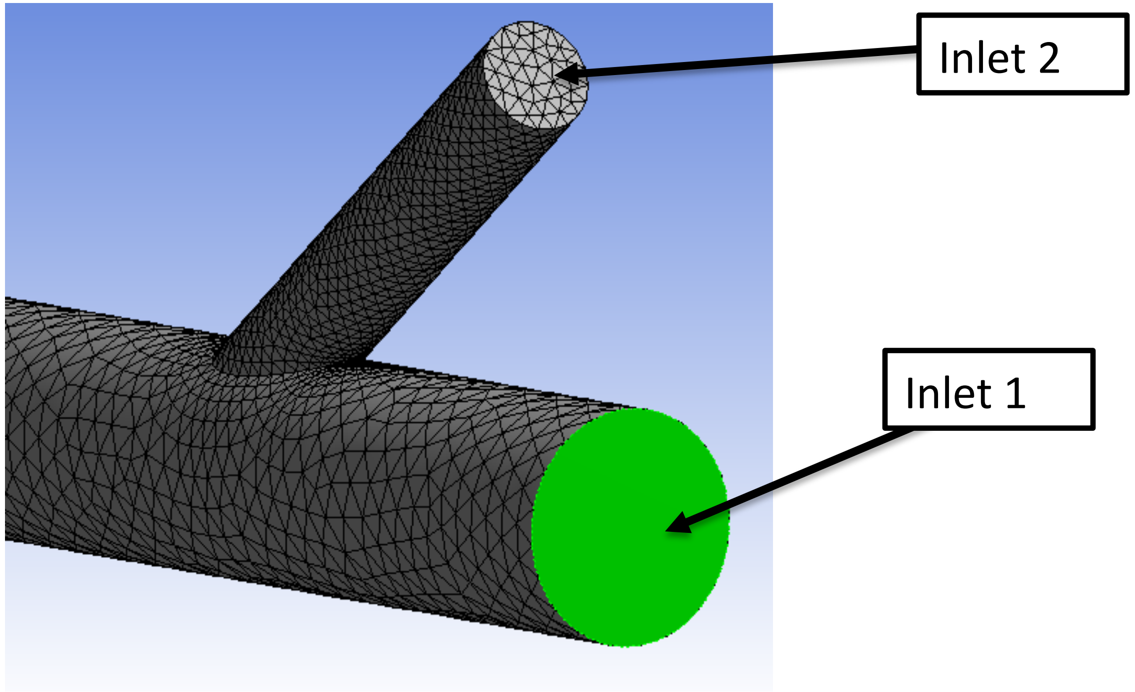
Solver Preference to CFX



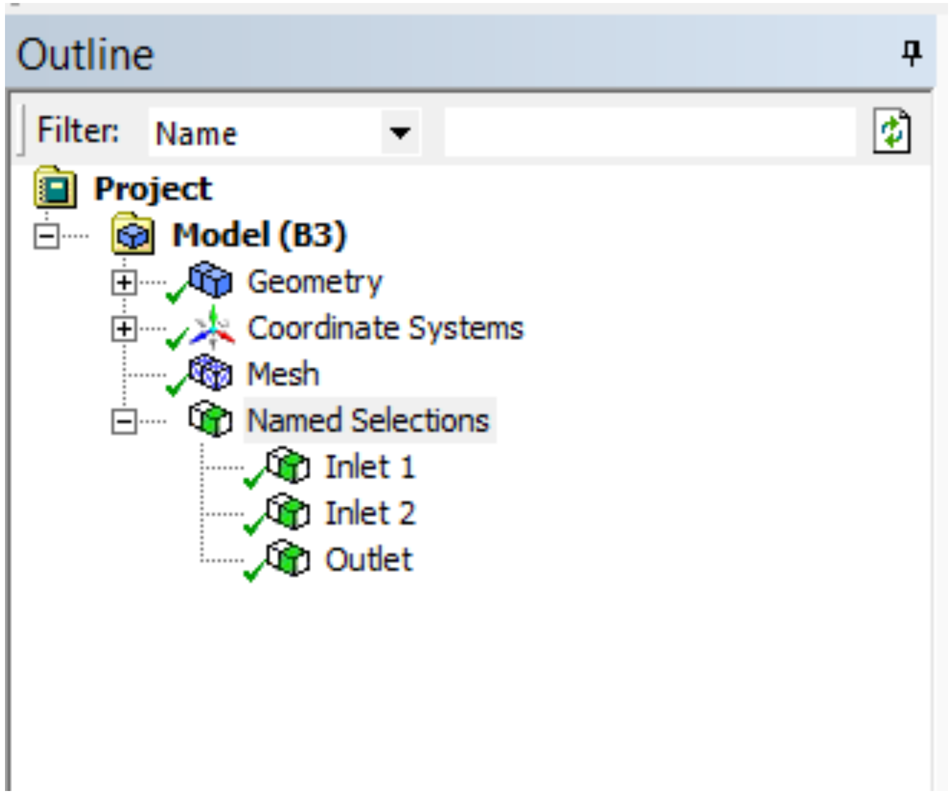
Details of "Mesh"

+ Display	
− Defaults	
Physics Preference	CFD
Solver Preference	CFX
Relevance	0
+ Sizing	
+ Inflation	
+ Patch Conforming Options	
+ Patch Independent Options	
+ Advanced	
+ Defeaturing	
− Statistics	
Nodes	9464
Elements	43114
Mesh Metric	None

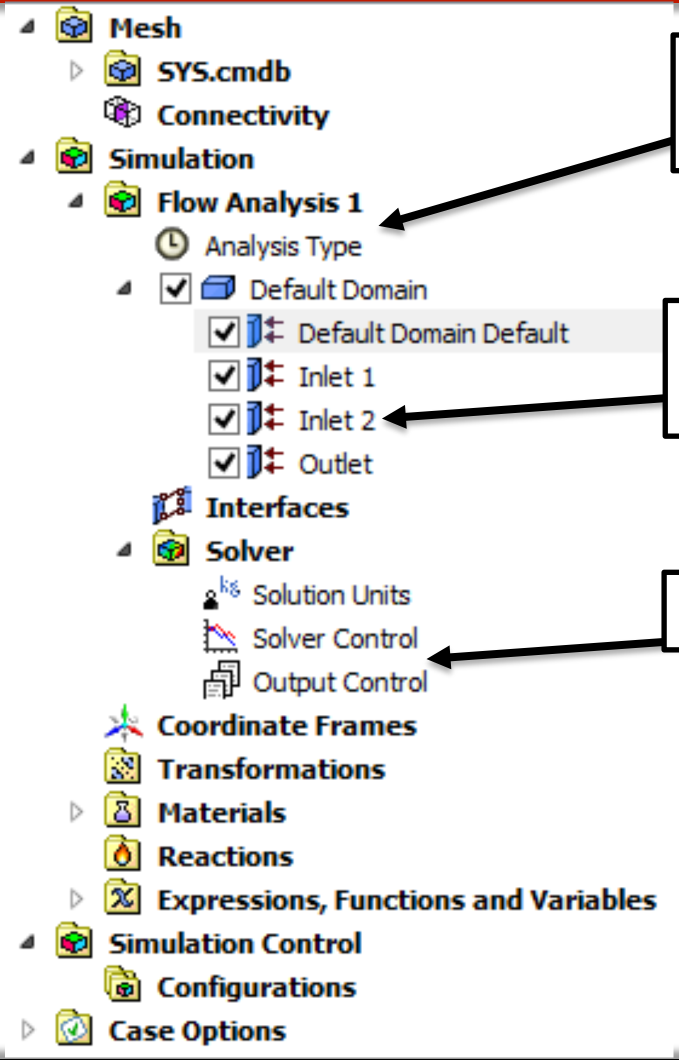
Mixing Pipe (Named Sections)



Naming sections now makes assigning boundary conditions easier.



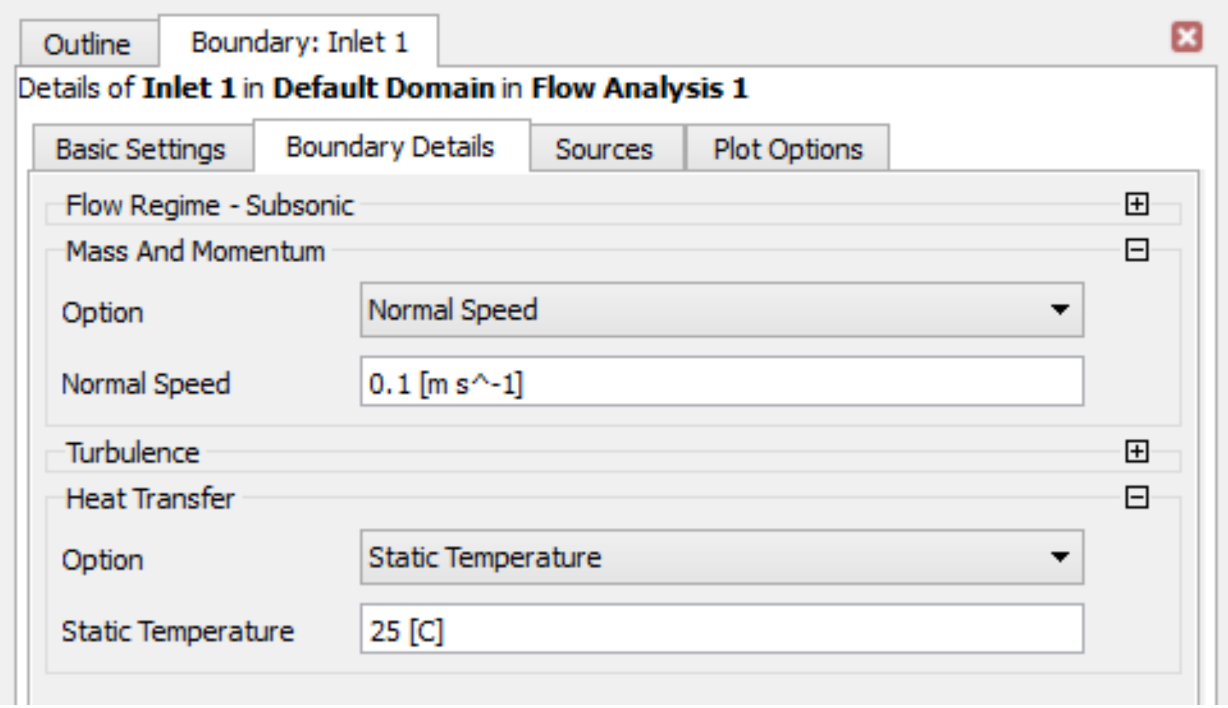
Mixing Pipe (CFX Setup)



Steady State or Transient

Boundary Conditions

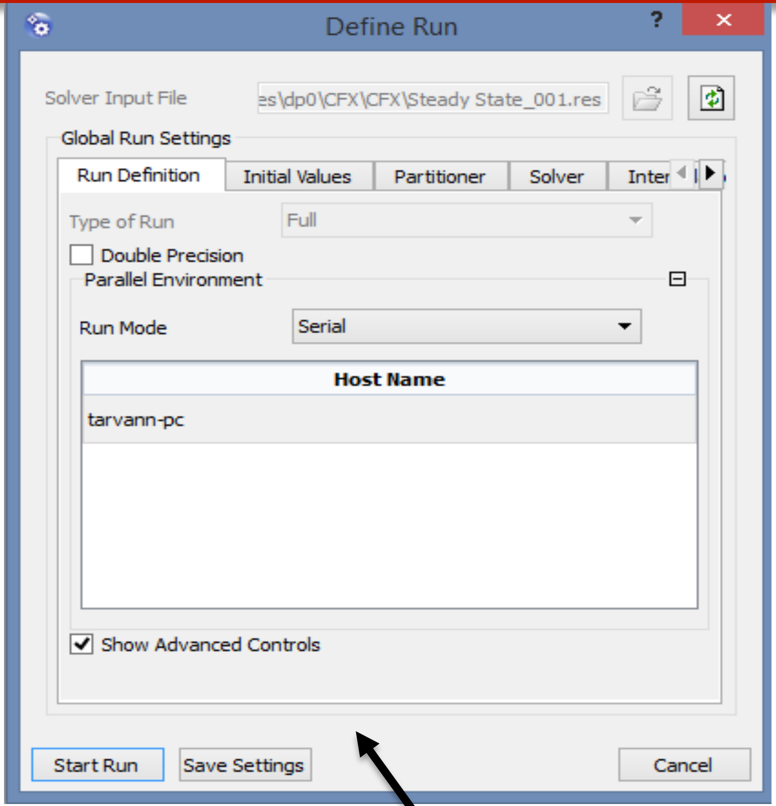
Solution Outputs



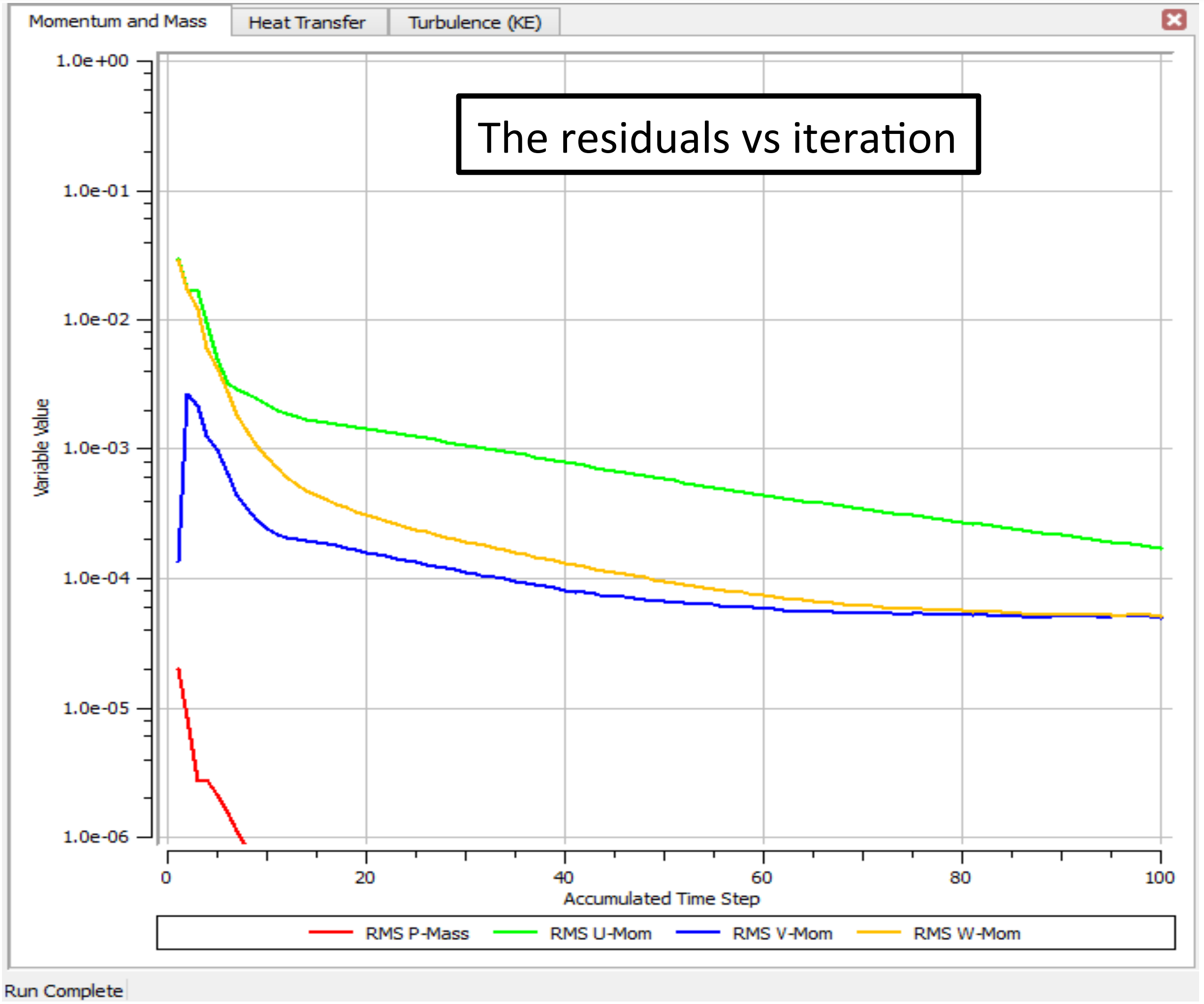
Inlet 2: Velocity (0.4 m/s) Temperature (50 ^{0}C)

Outlet: Pressure (100 kPa)

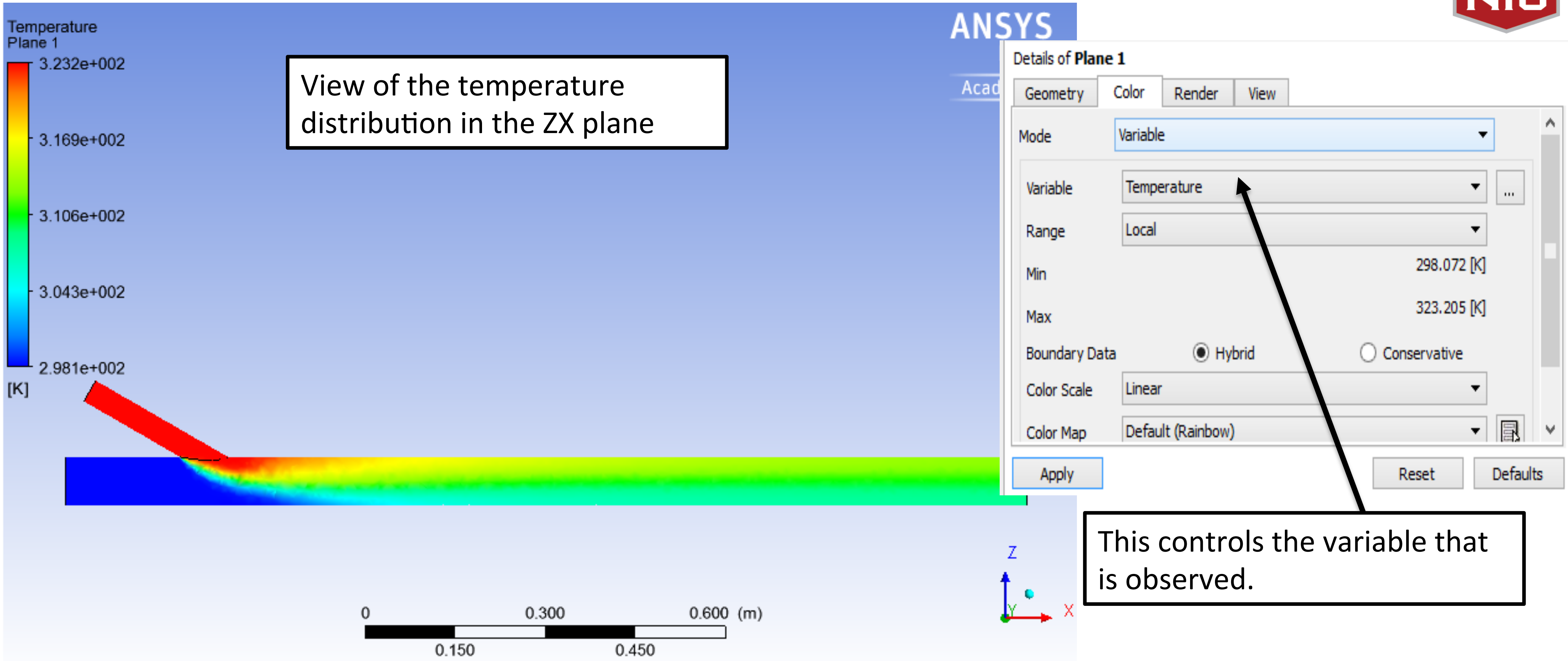
Mixing Pipe Tutorial (CFX Solver)



Solution Manager gives several options to help optimize run time.



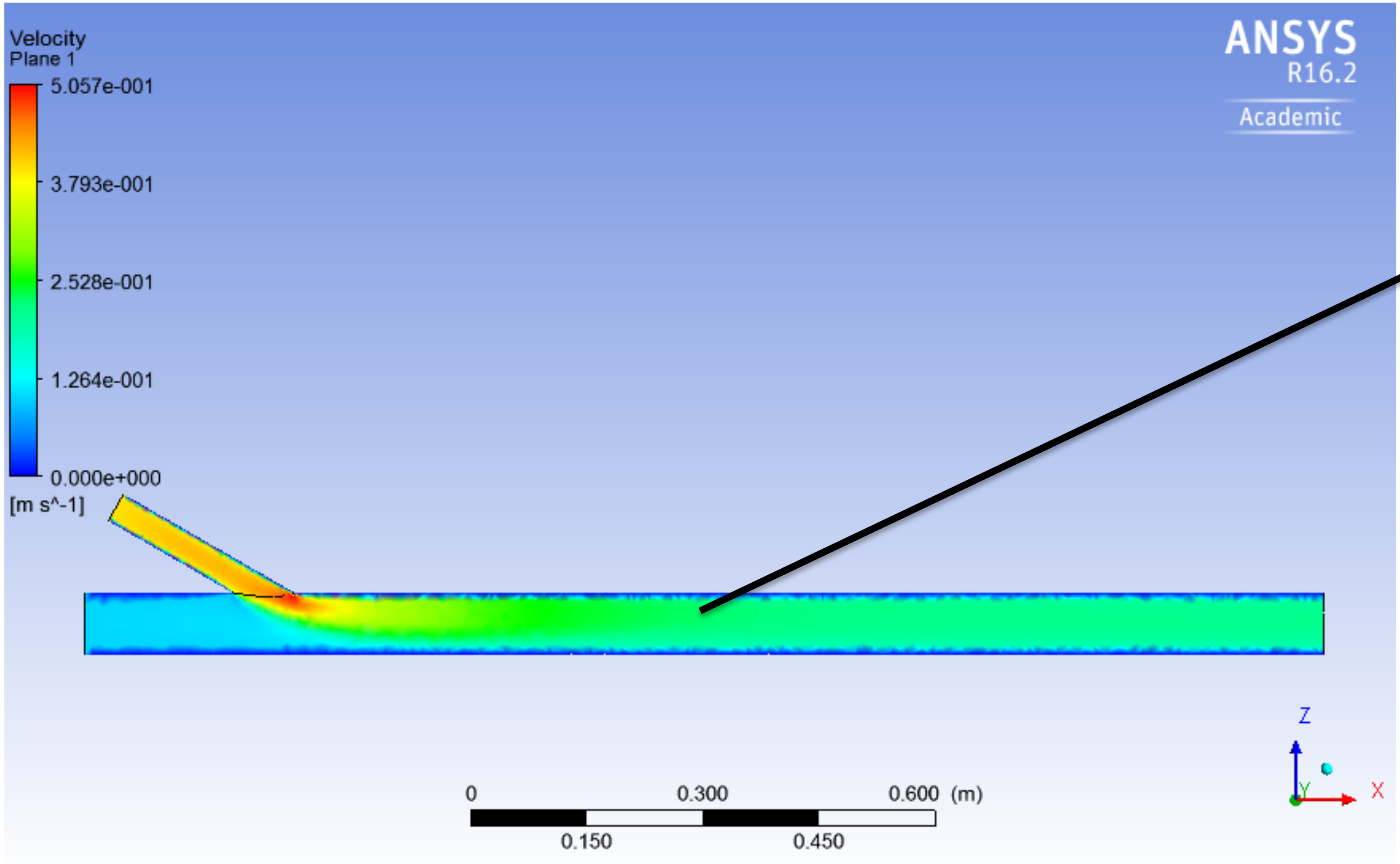
Mixing Pipe (Results)

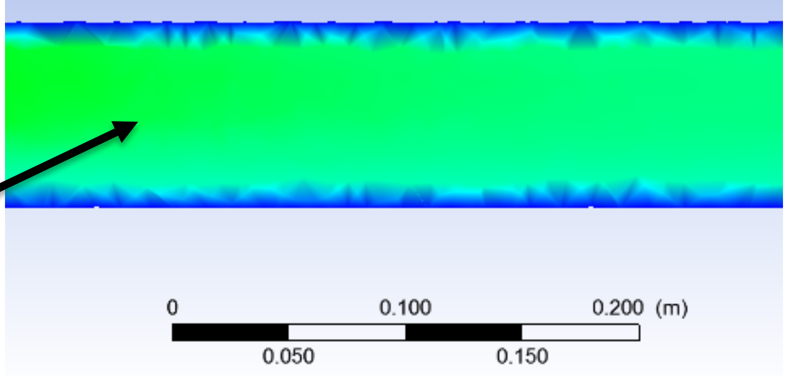


View of the temperature distribution in the ZX plane

This controls the variable that is observed.

Mixing Pipe (Results)





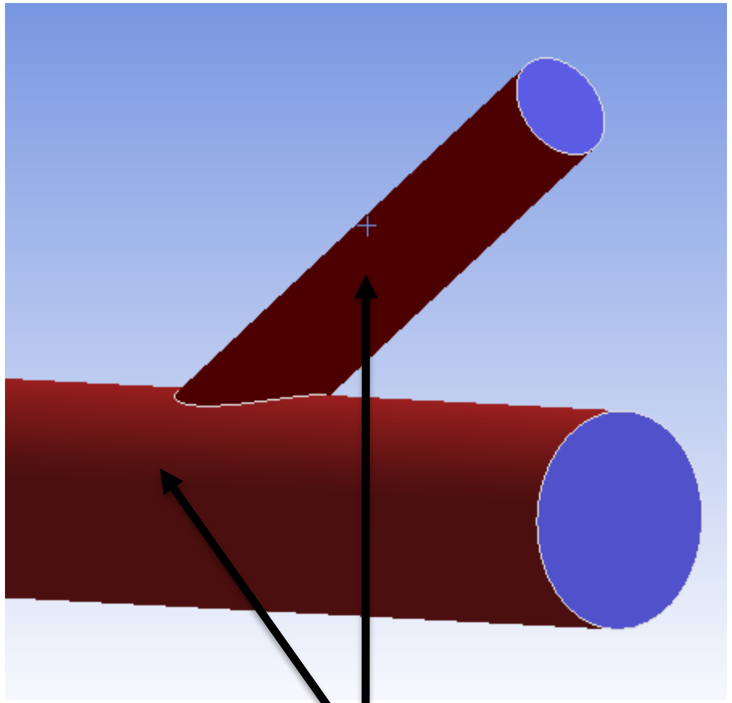
The velocity distribution is very jagged near the wall. Perhaps the model can be refined to improve the solution.

Mixing Pipe (Improvements)

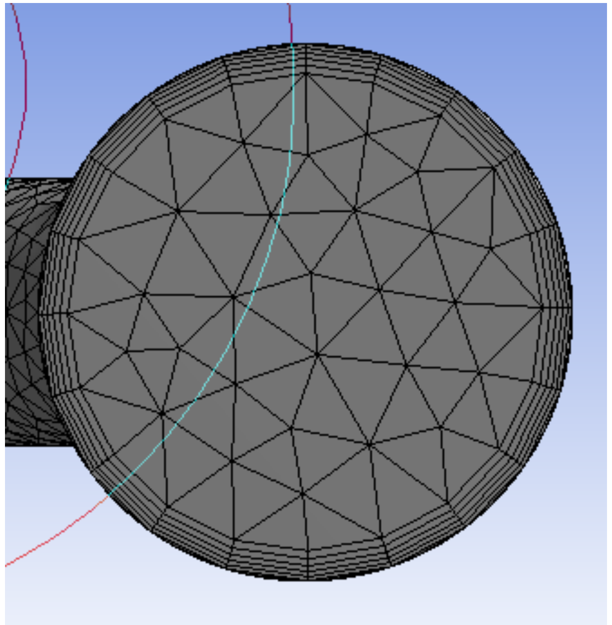
Details of "Inflation" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	2 Faces
Inflation Option	Smooth Transition
Transition Ratio	0.1
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre

This determines how much inflation there will be.

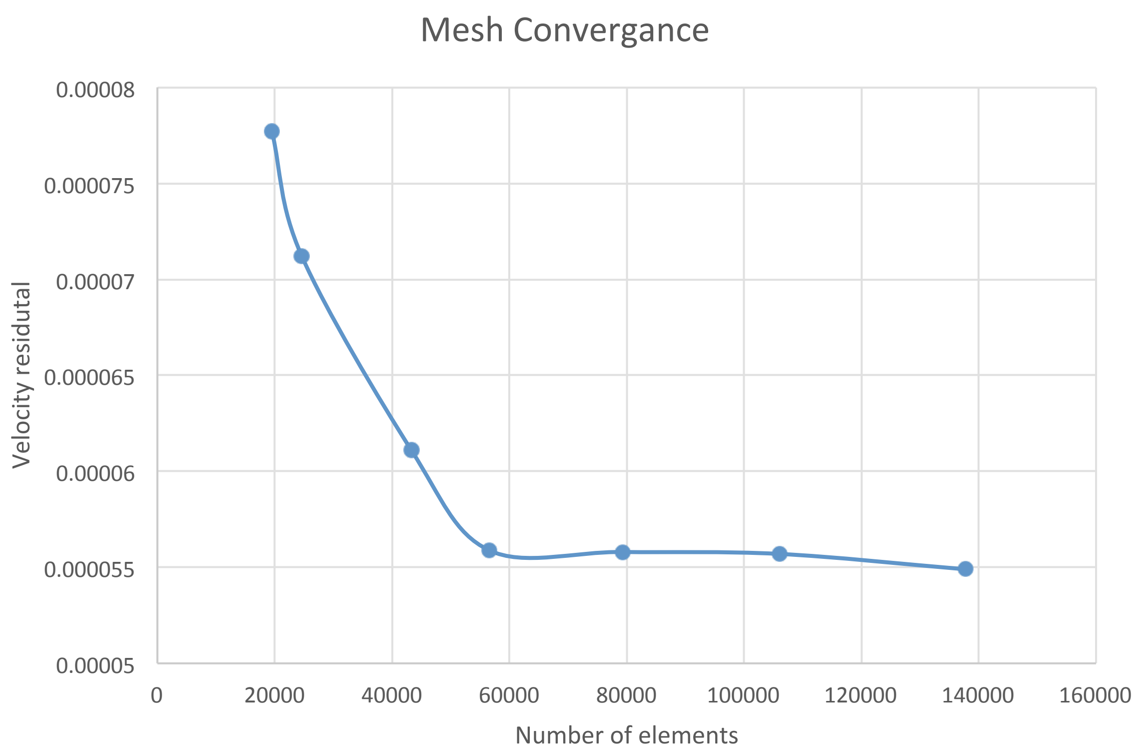


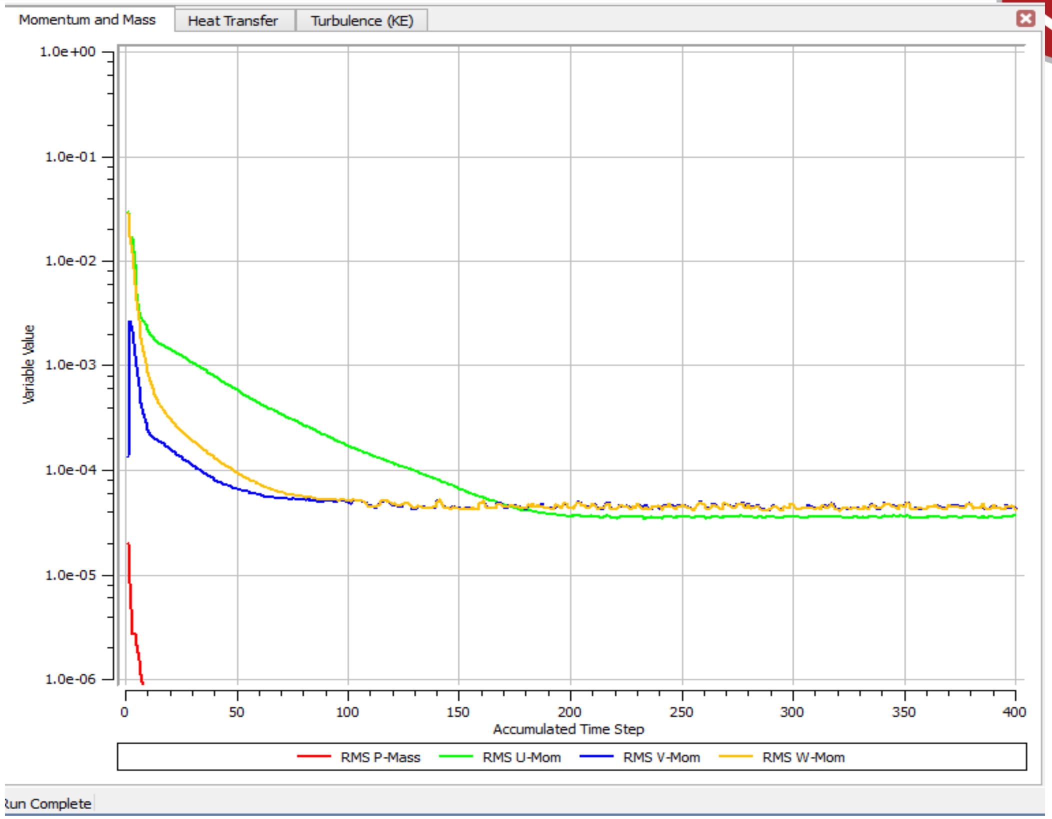
These are the 2 surface boundaries.



This greatly increases density at the wall.

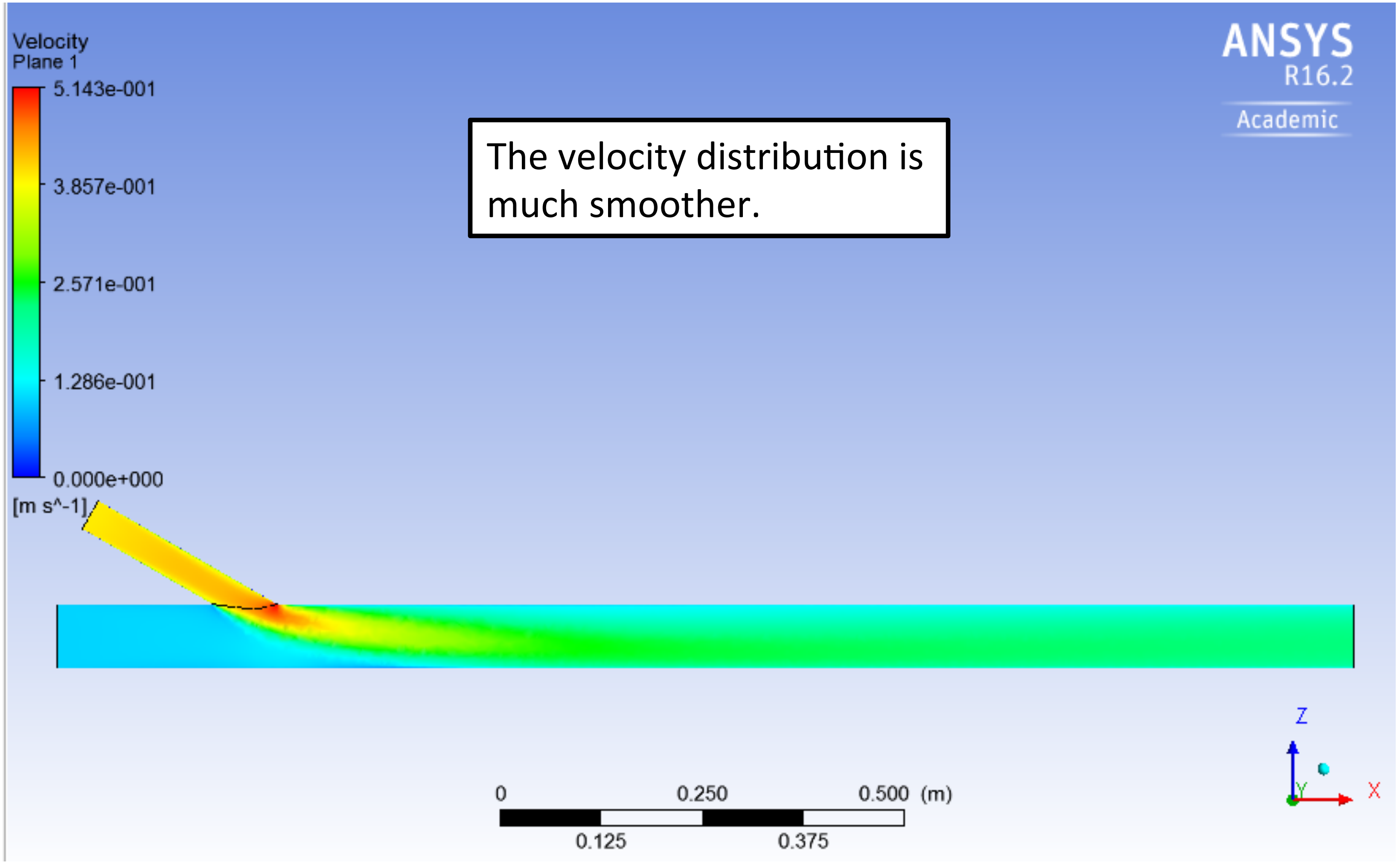
Mixing Pipe (Improvements)





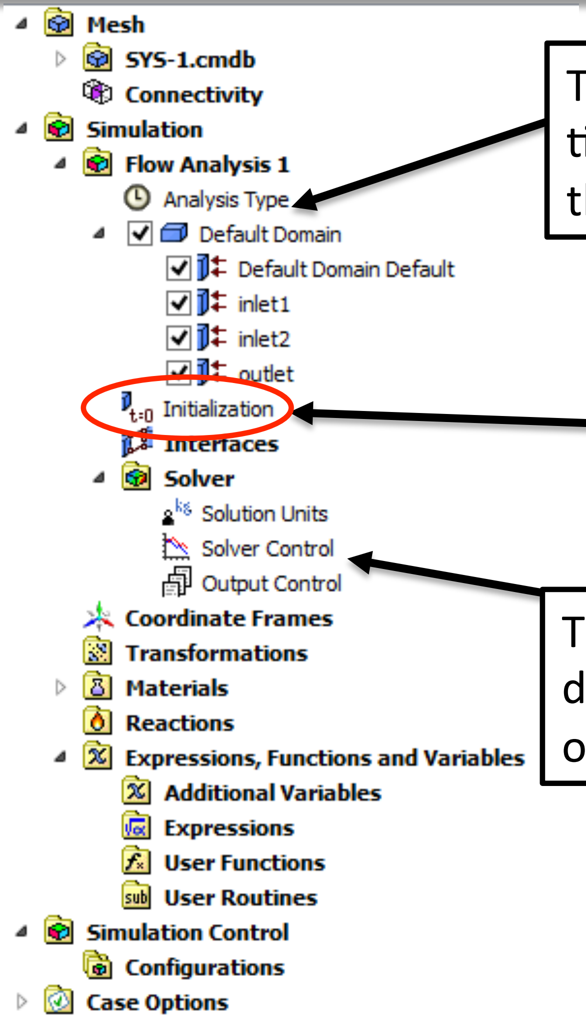
Changing the size of the mesh affects when the residual graph will converge. Run the simulation for more iterations will also help increase the accuracy of the solution.

Mixing Pipe (Improvements)



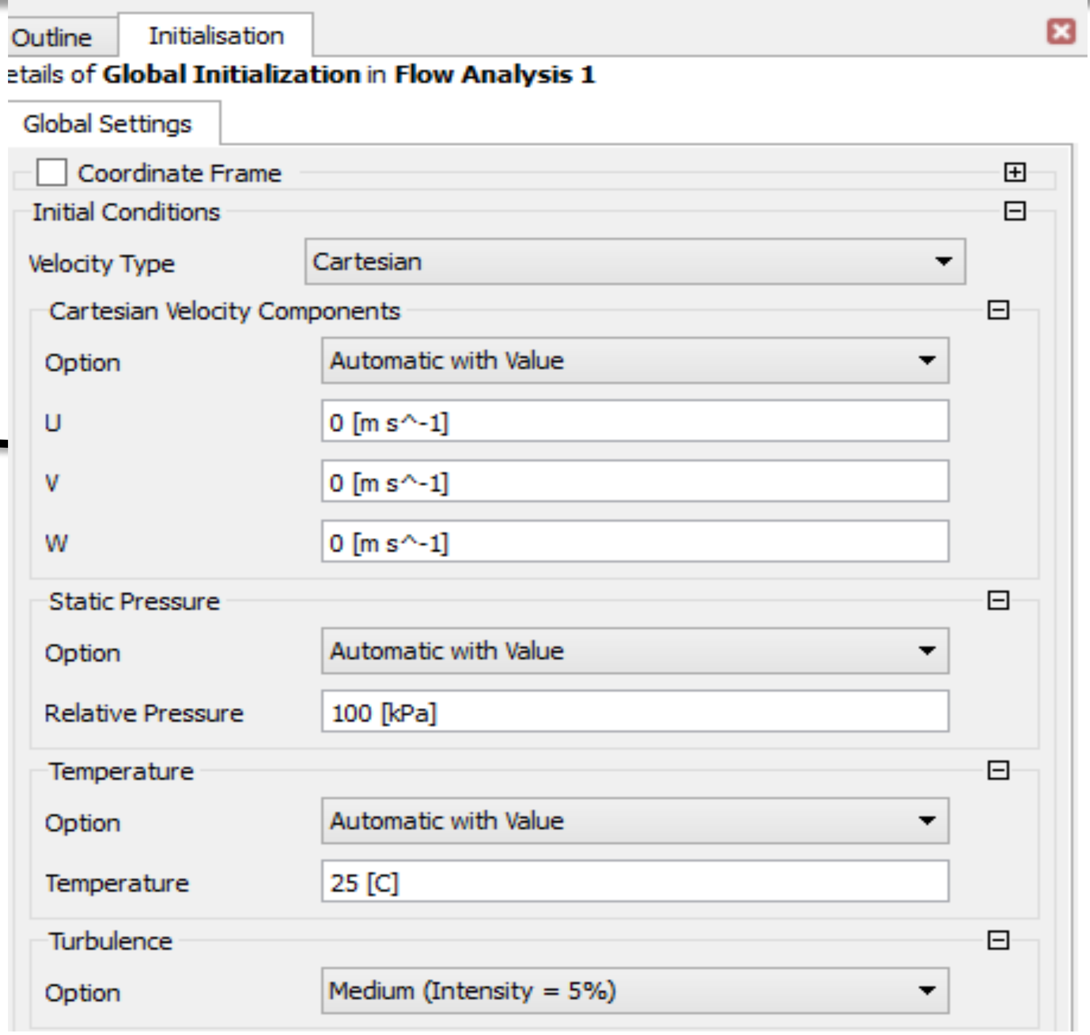
The velocity distribution is much smoother.

Mixing Pipe (Transient)

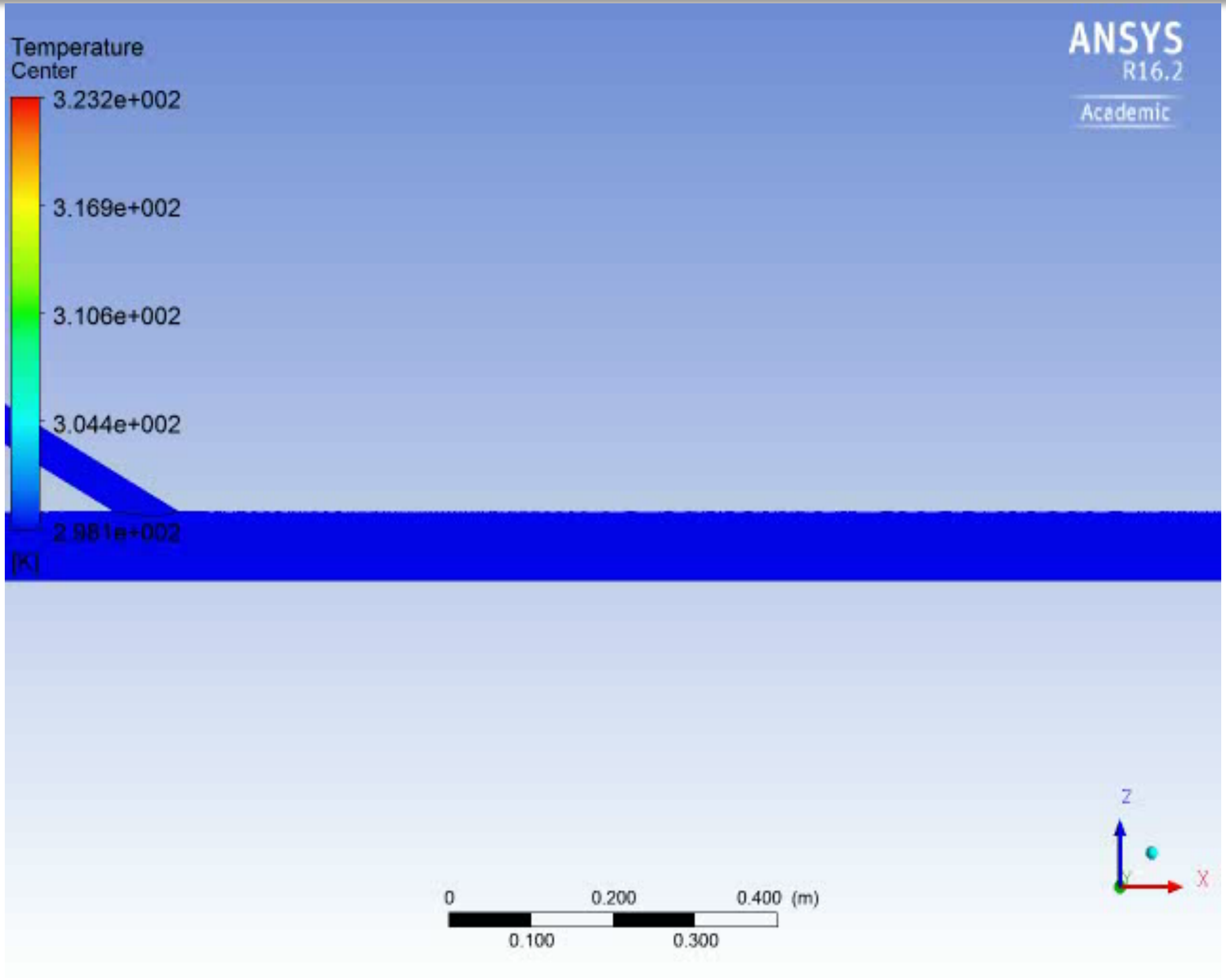


This controls the total time of the simulation and the time step.

This controls how much data is recorded and how often.



Video

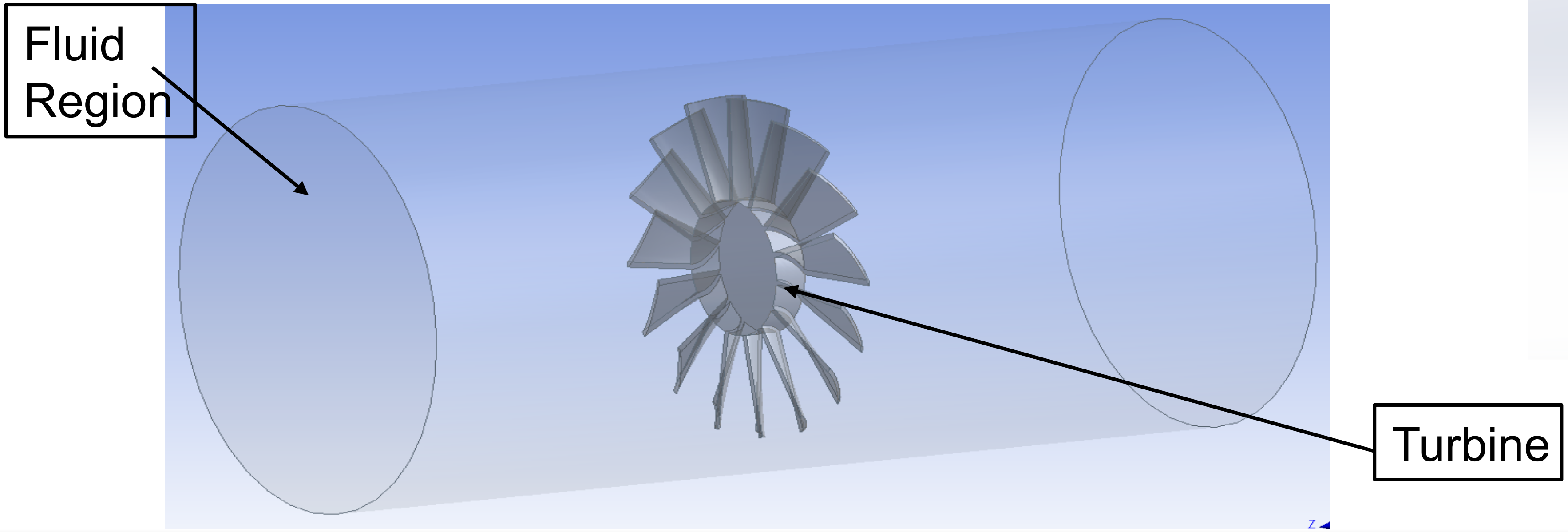

Temperature
Center
3.232e+002
3.169e+002
3.106e+002
3.044e+002
2.981e+002
[K]
ANSYS
R16.2
Academic
0
0.200
0.400 (m)
0.100
0.300
Z
X

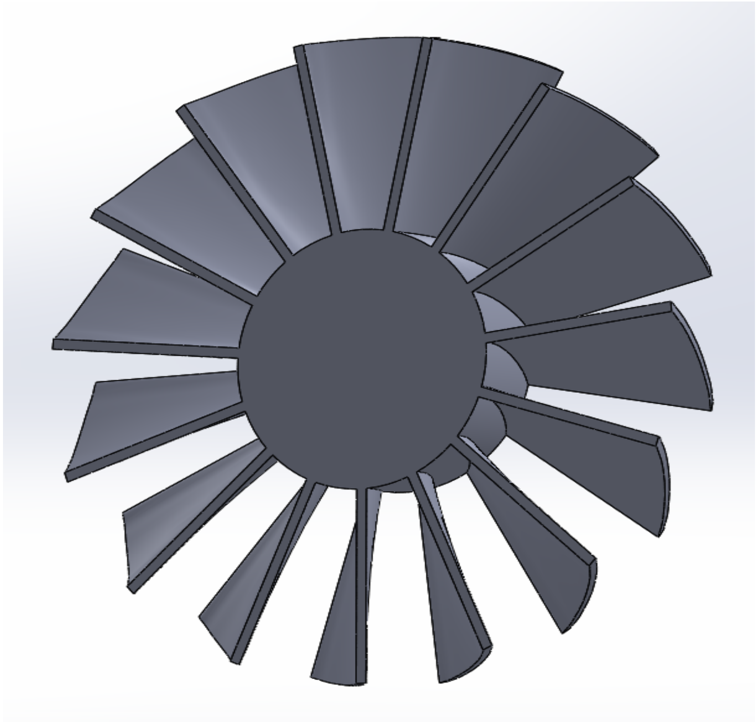
Part 2 – Single Stage Low Pressure Turbine

- Problem Statement – To study changes in physical properties of steam as it is driven through turbine using steady and transient conditions.

Geometry Details

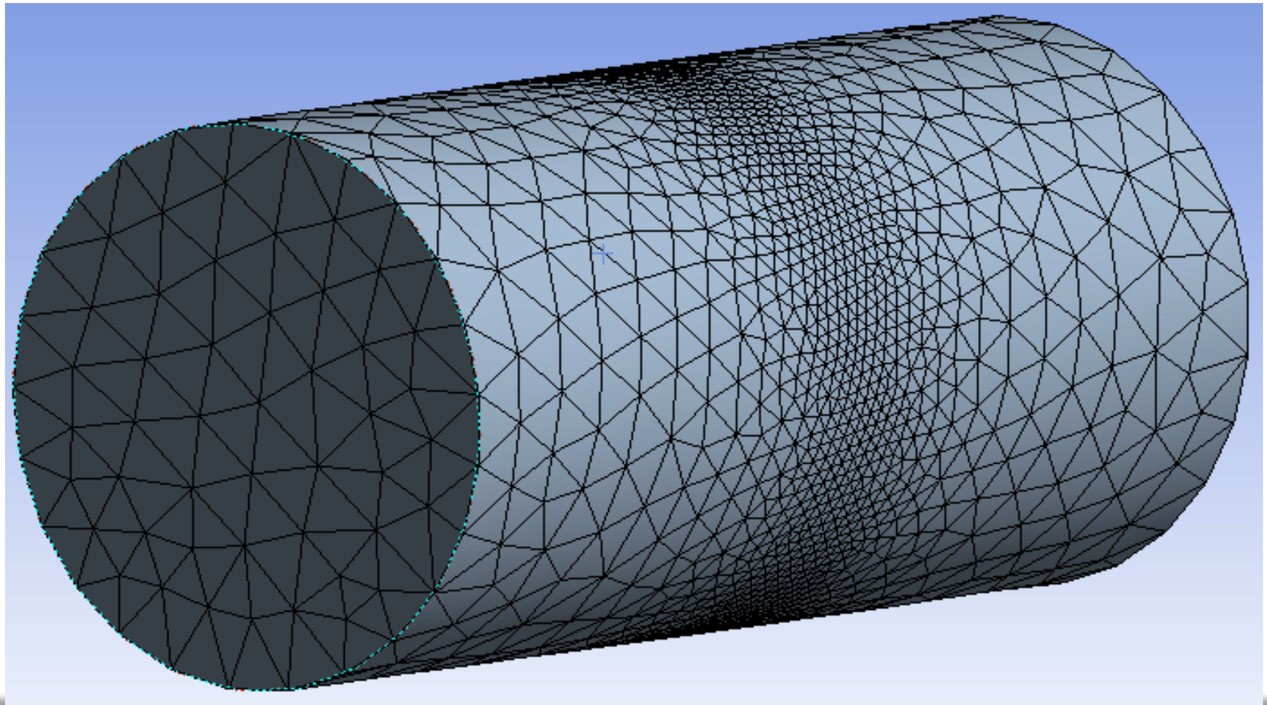
- Solidworks used for modelling turbine.
- No. of blades – 15
- Material – Steel
- Overall Dimension–Φ4x6.2m (Including enclosure)

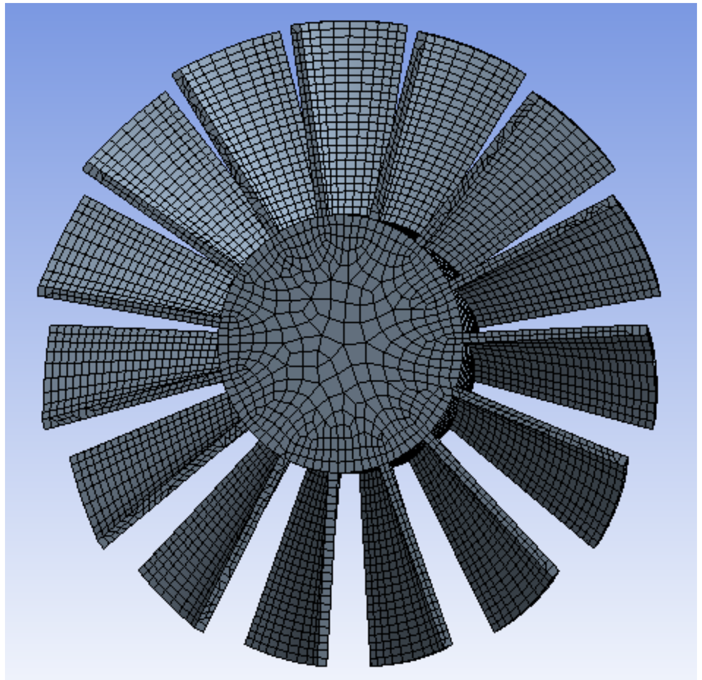




Mesh Details

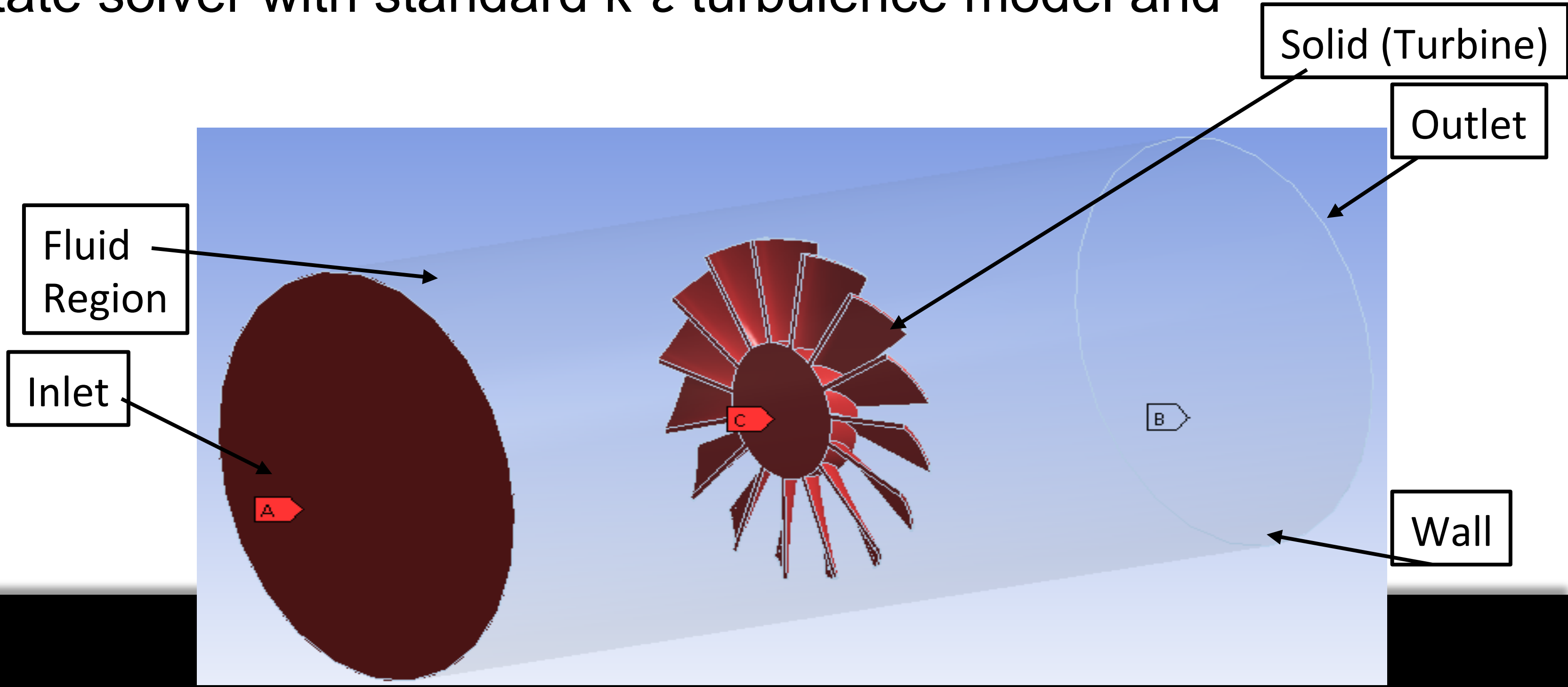
- Combination of Tetrahedron and Hexahedron.
- Coarse mesh due to limitation of elements (Limit 510,000)
- No. of elements – 345,244. No. of nodes – 79,316



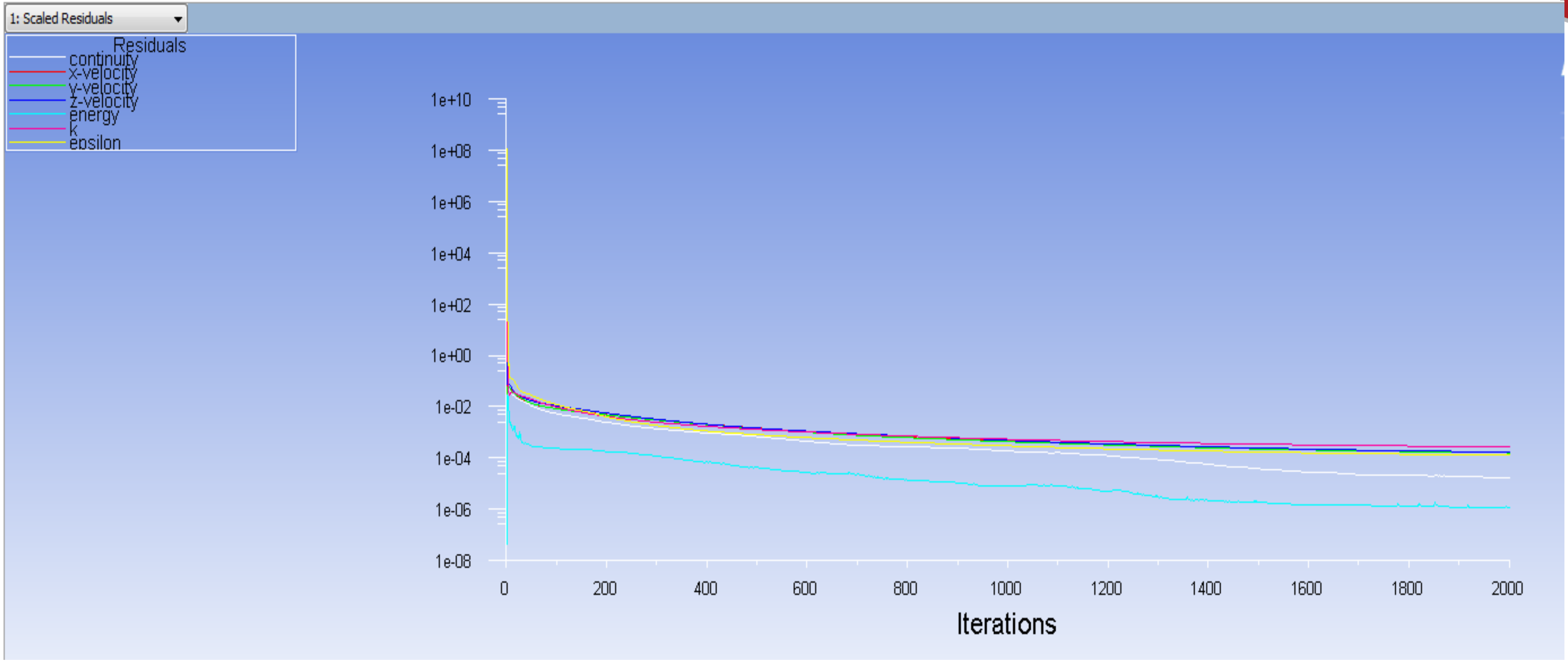


Boundary Conditions and solver details

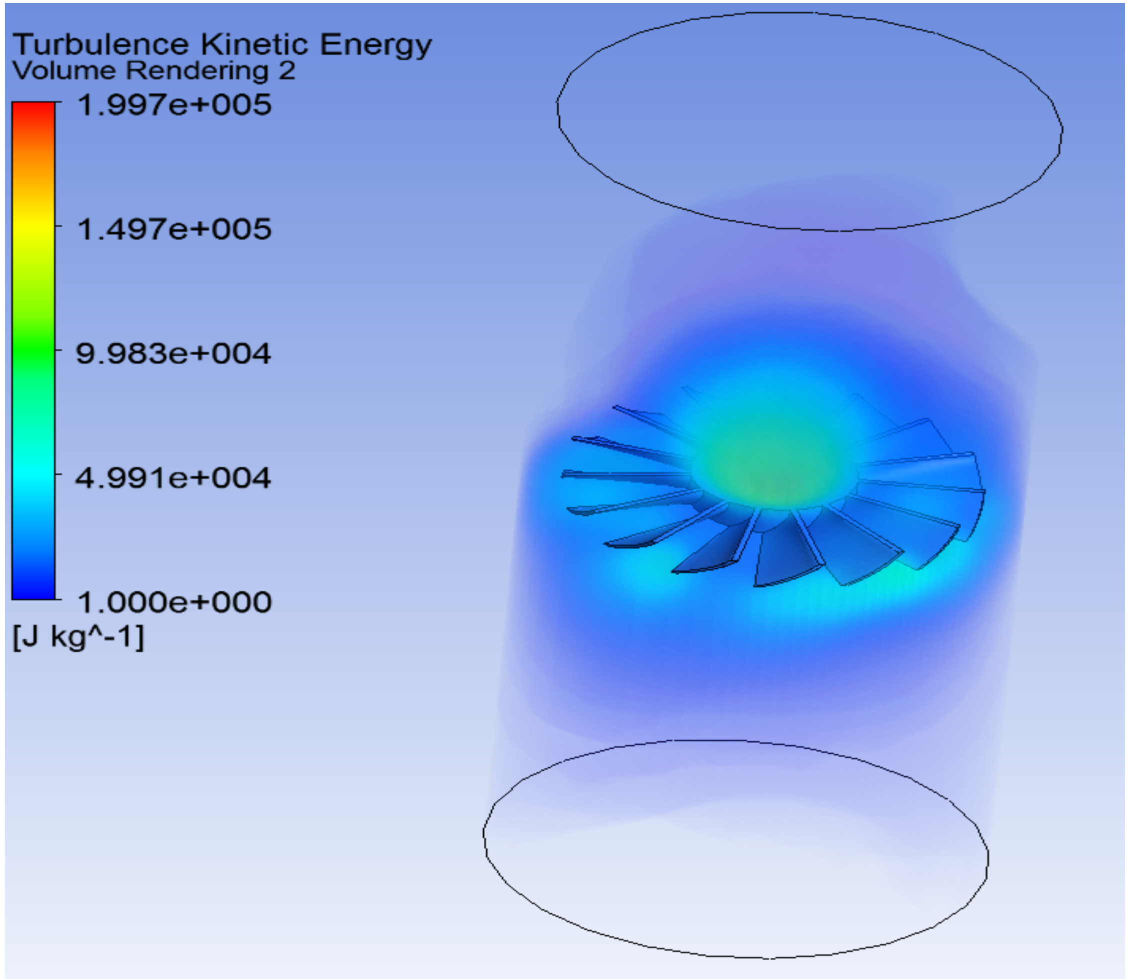
- Fluid – Steam
- Inlet Conditions – Velocity Inlet -1.5 MPa, 623K, 10 m/s.
- Outlet Conditions – Pressure Outlet – 5KPa, 305K,
- Wall – No-slip, Stationary wall.
- Pressure-based, steady state solver with standard k-ε turbulence model and energy equation.
- SIMPLE Scheme

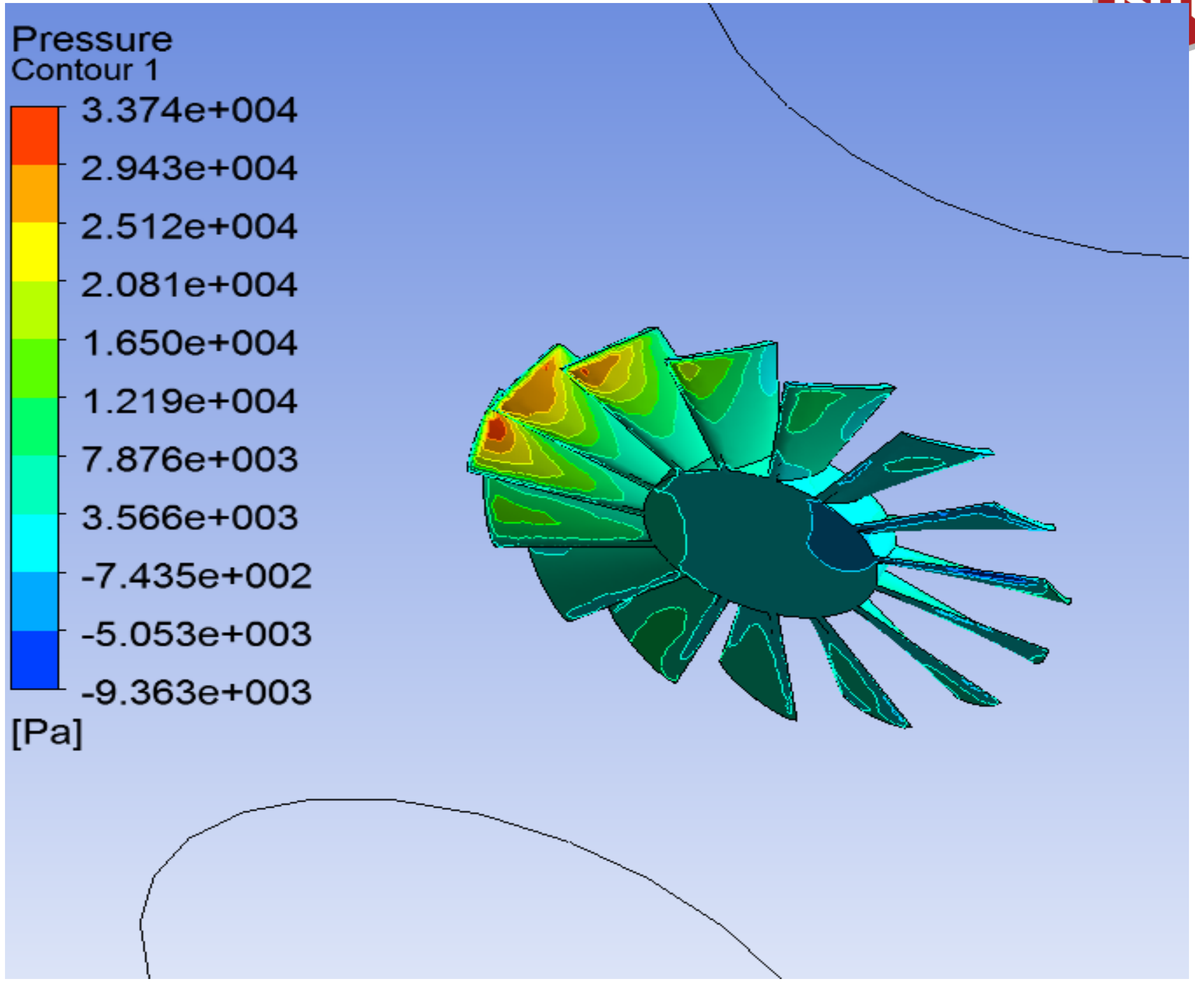


Results - Steady Condition

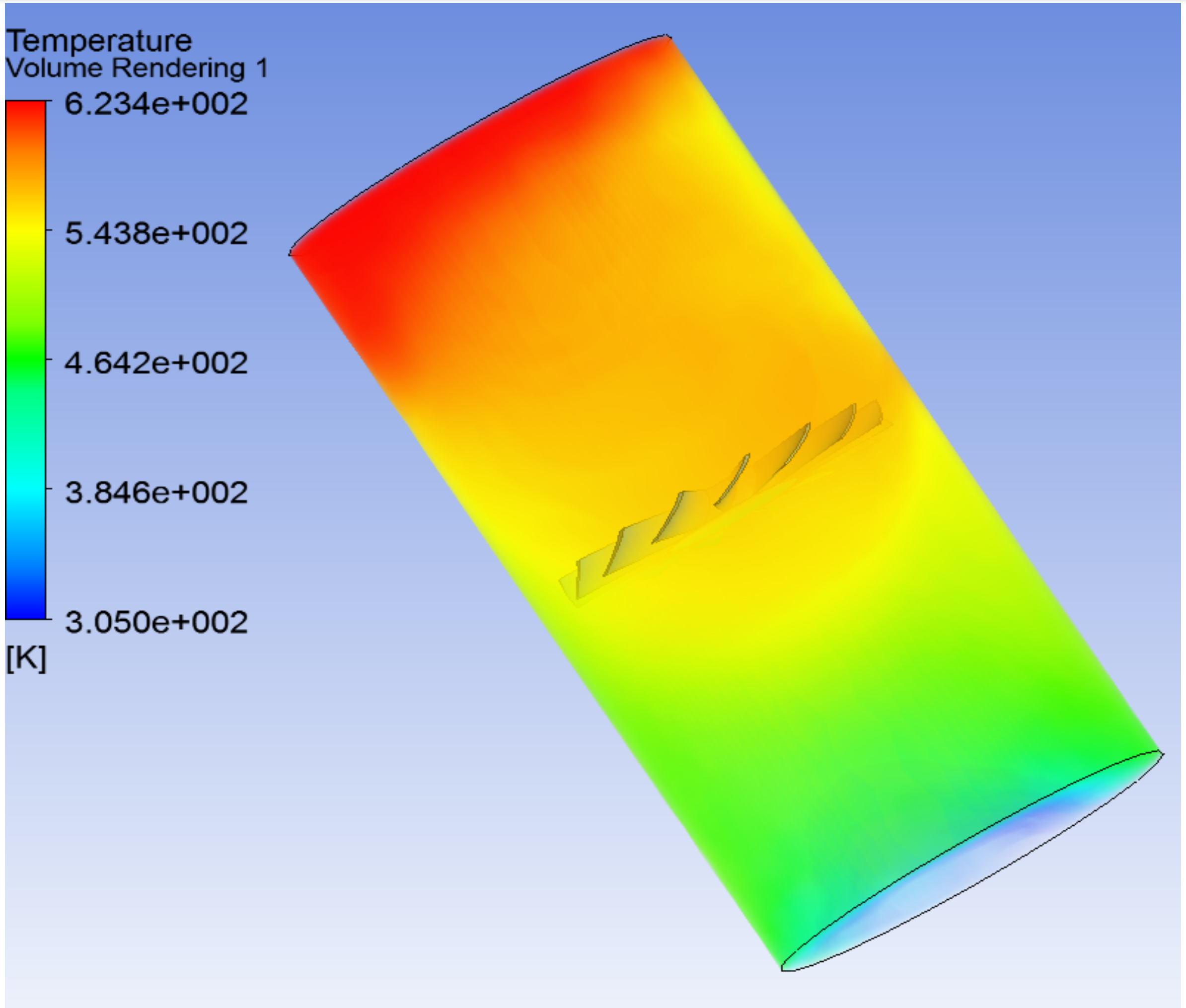


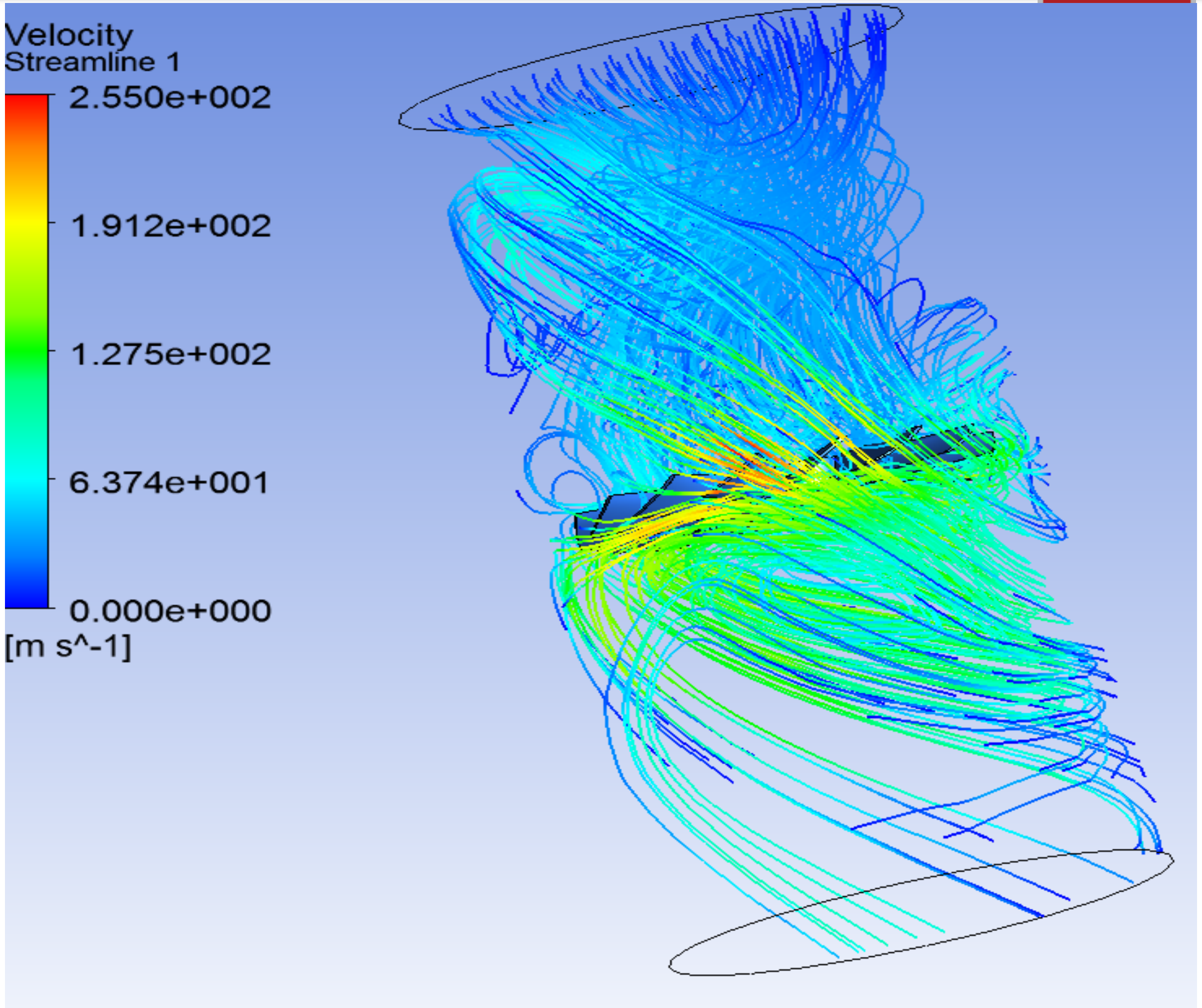
Results - Steady Condition



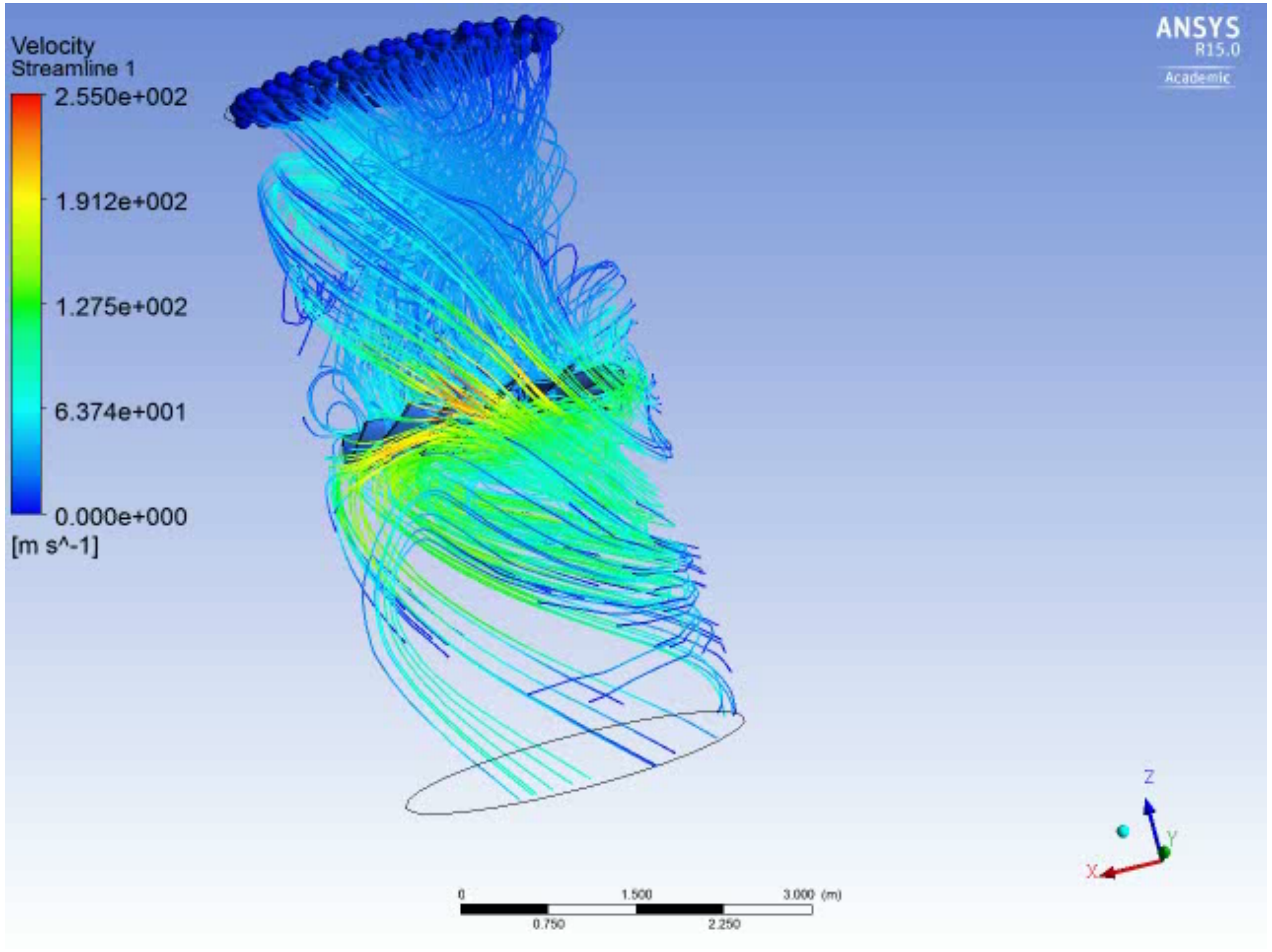


Results - Steady Condition



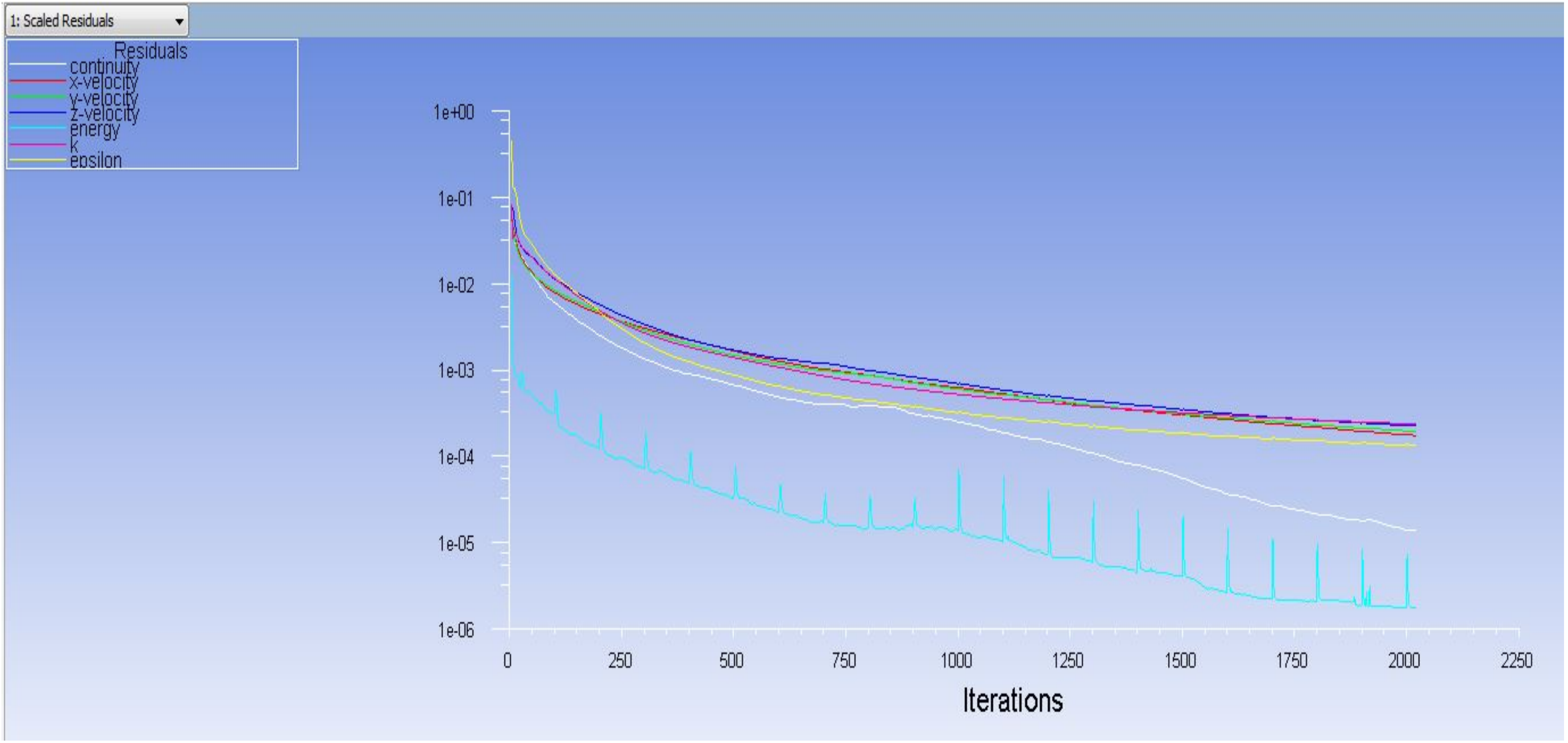


Video

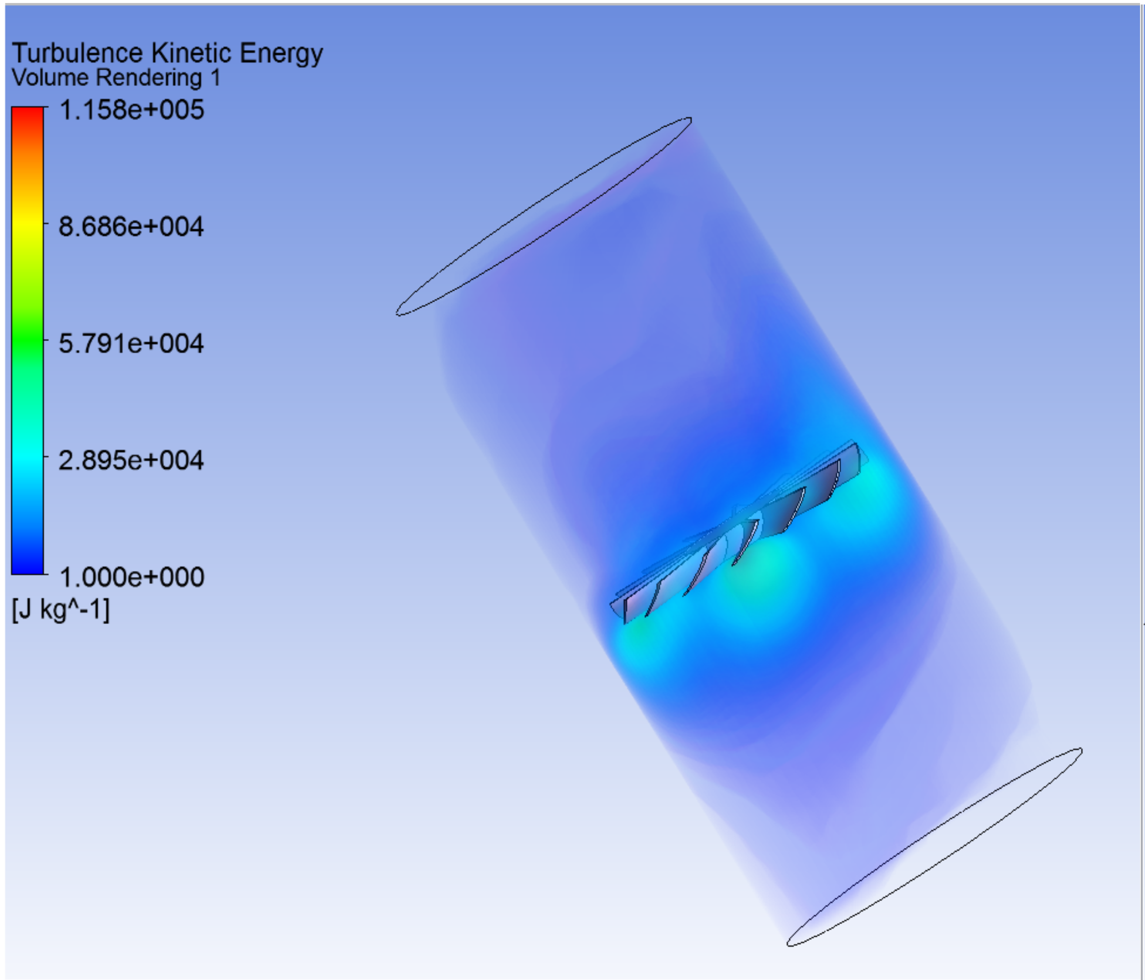


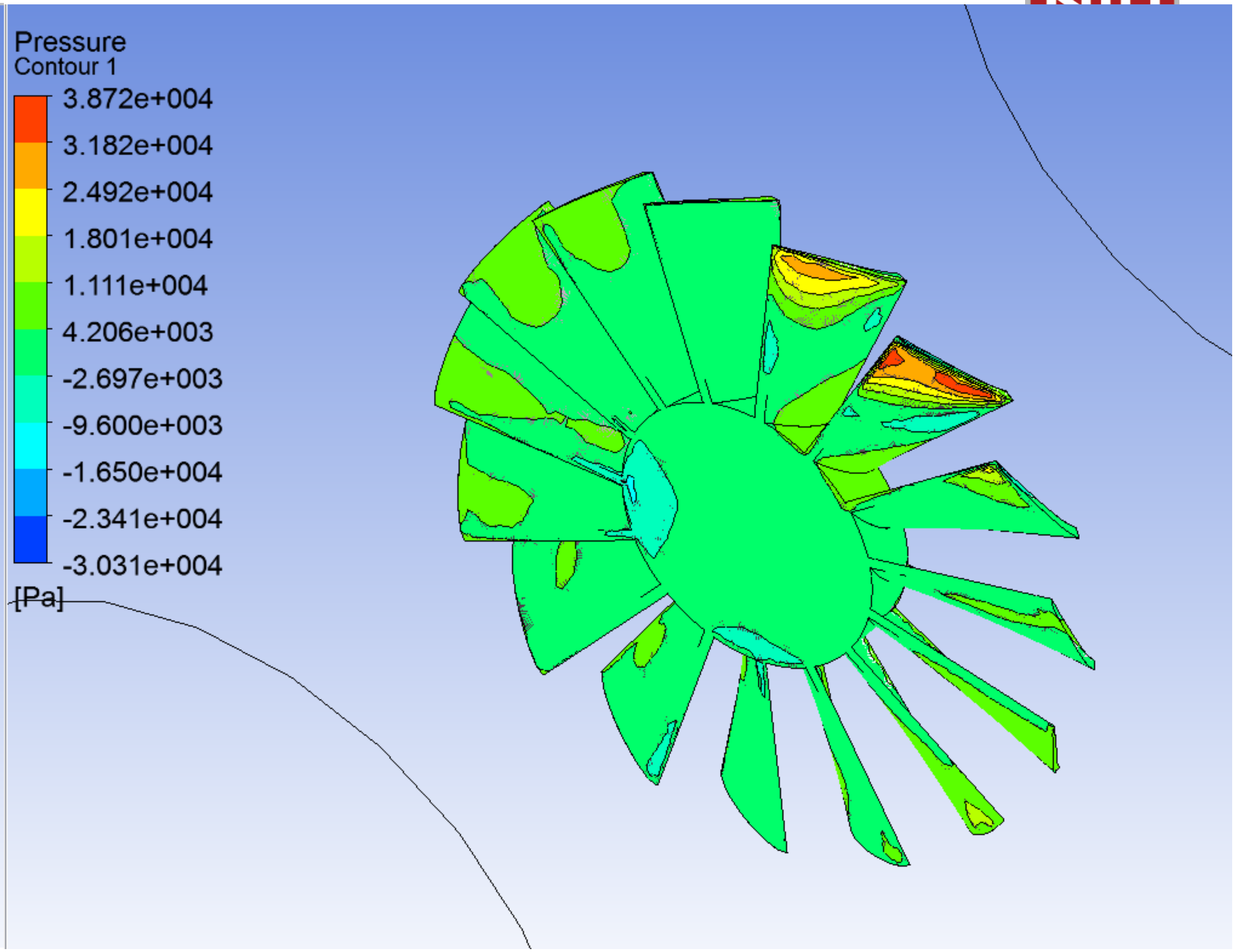
Velocity
Streamline 1
2.550e+002
1.912e+002
1.275e+002
6.374e+001
0.000e+000
[m s^-1]
ANSYS
R15.0
Academic
0
1.500
3.000 (m)
0.750
2.250
Z
Y
X

Results – Transient Condition

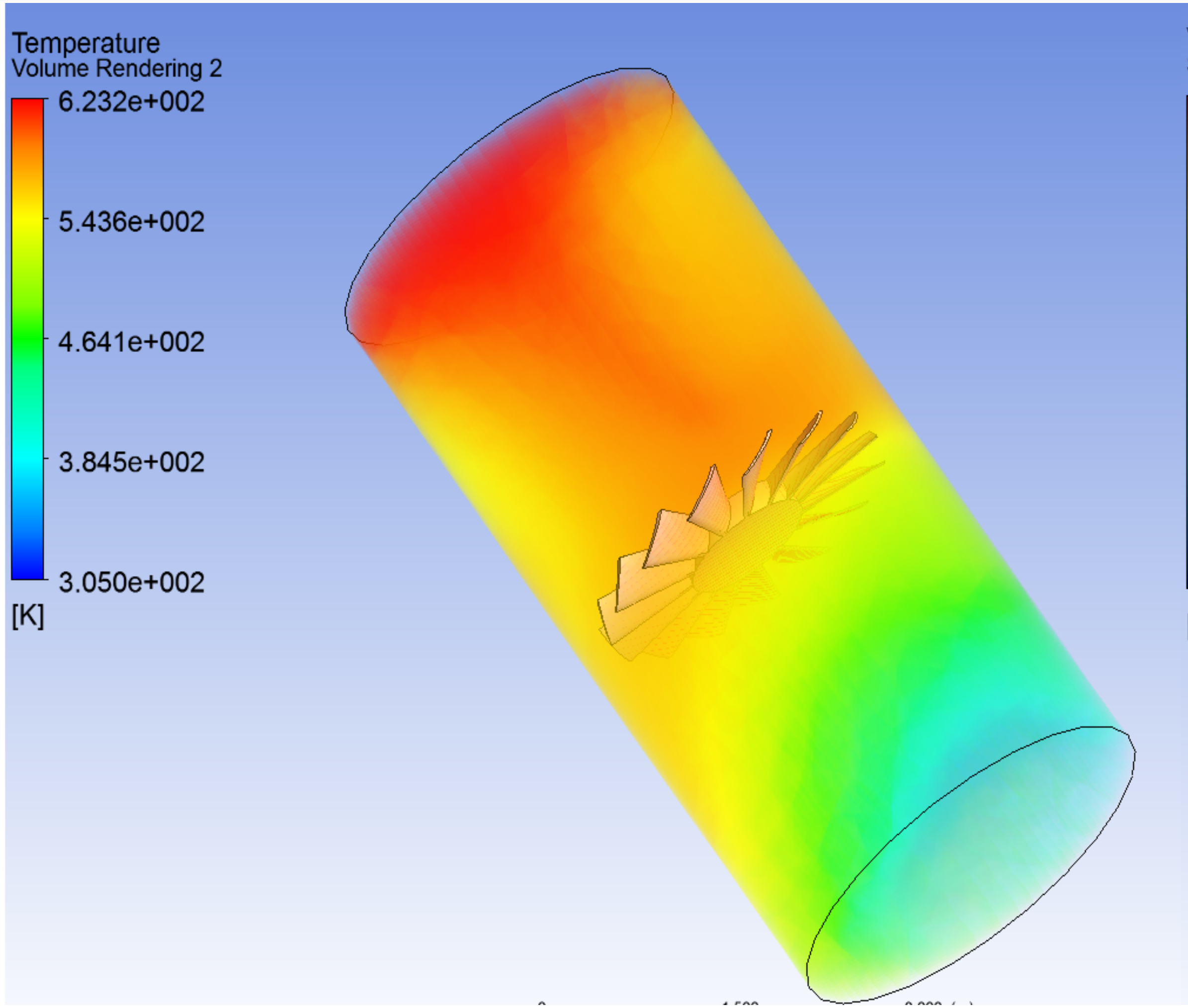


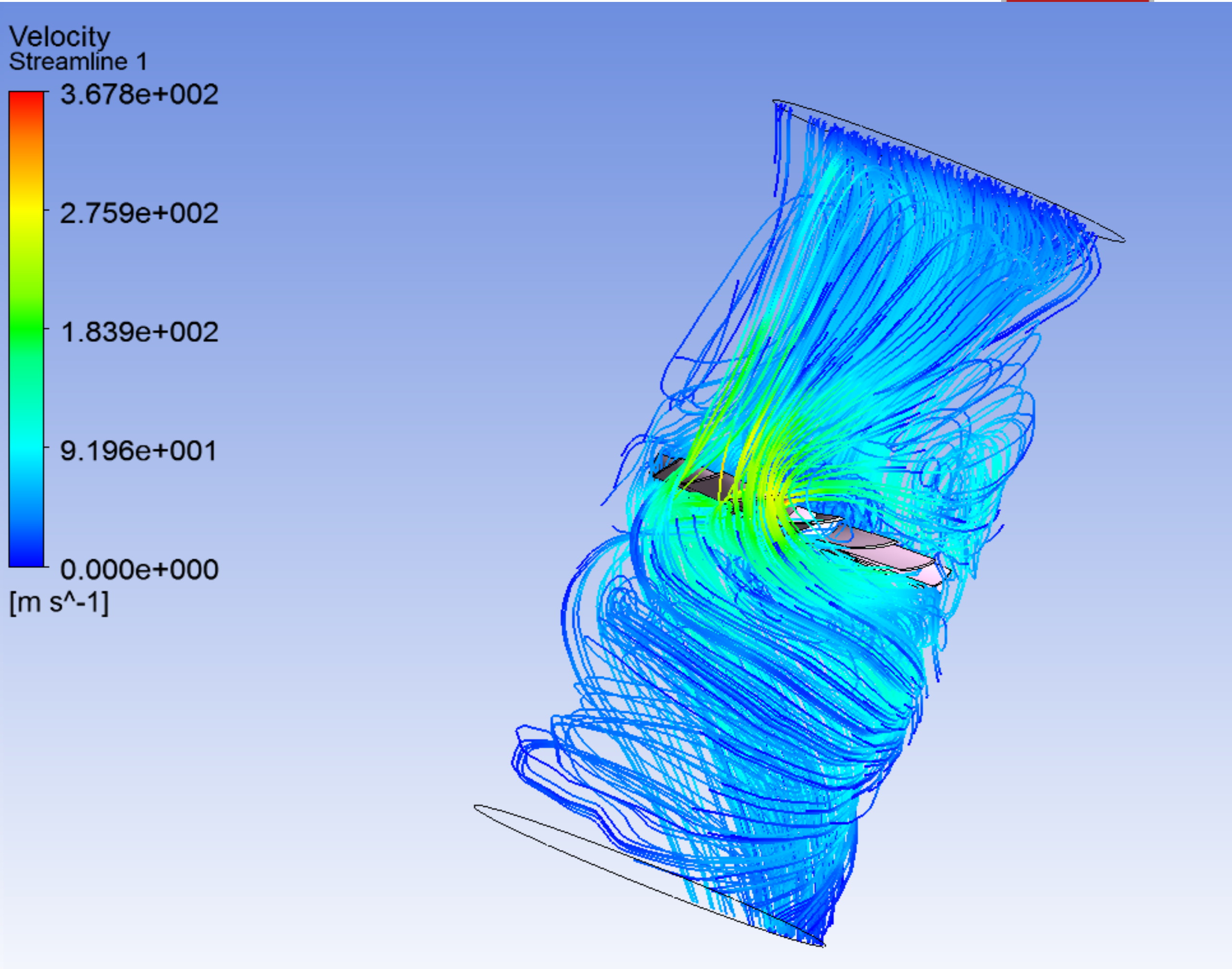
Results – Transient Condition



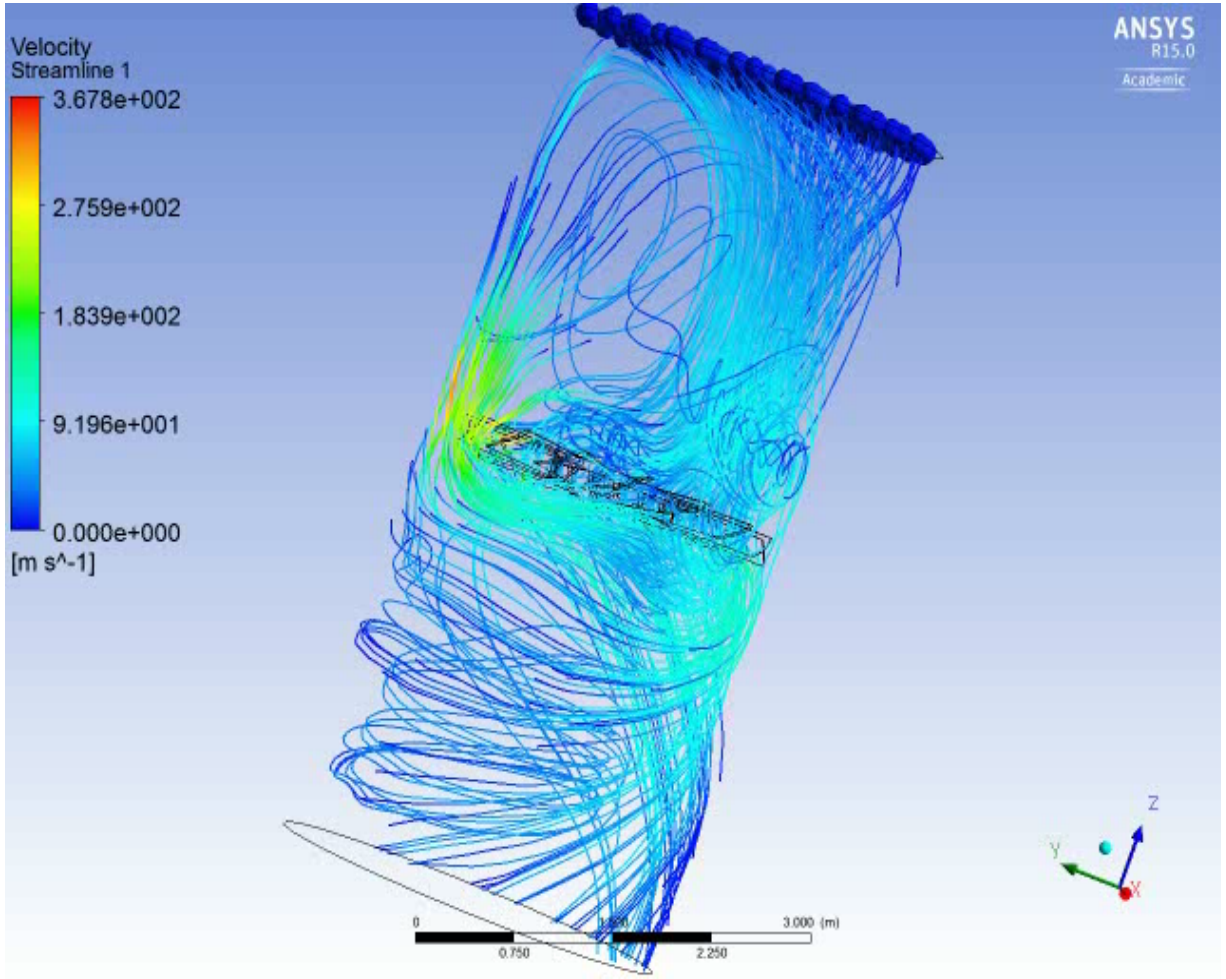


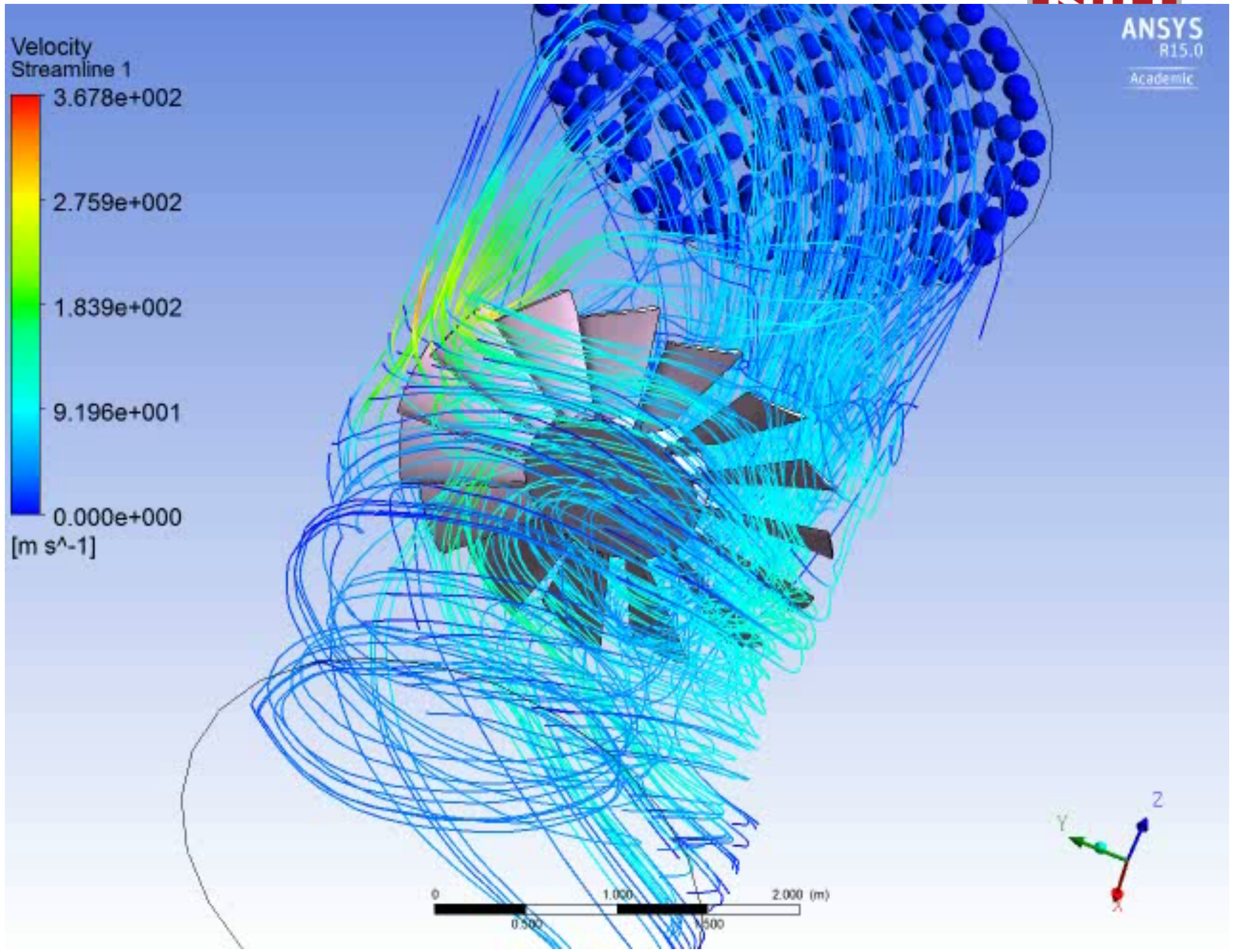
Results – Transient Condition



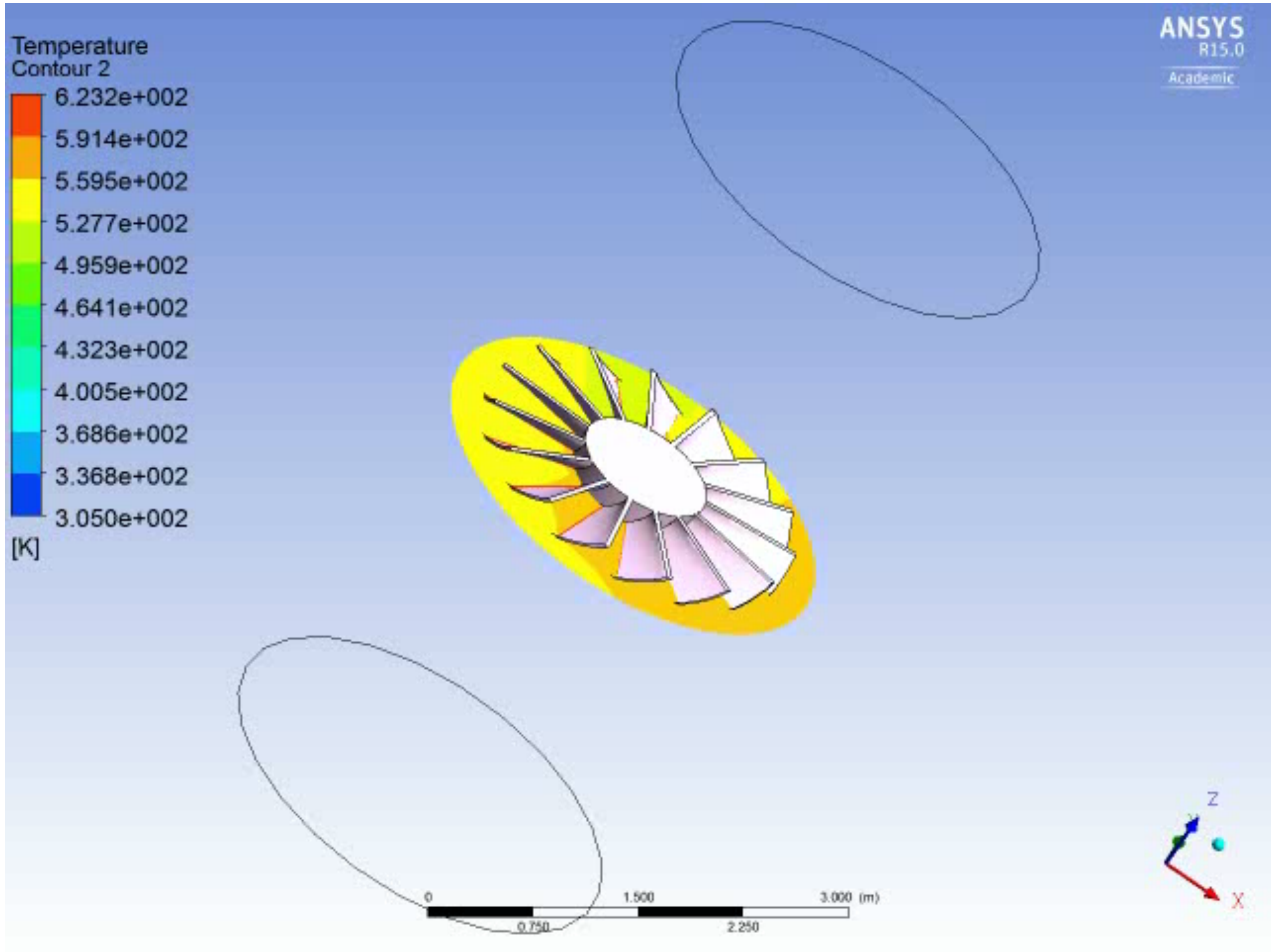


Videos


Velocity
Streamline 1
3.678e+002
2.759e+002
1.839e+002
9.196e+001
0.000e+000
[m s^-1]
ANSYS
R15.0
Academic


Velocity
Streamline 1
3.678e+002
2.759e+002
1.839e+002
9.196e+001
0.000e+000
[m s^-1]
ANSYS
R15.0
Academic

Videos



ANSYS
R15.0
Academic
Temperature
Contour 2
6.232e+002
5.914e+002
5.595e+002
5.277e+002
4.959e+002
4.641e+002
4.323e+002
4.005e+002
3.686e+002
3.368e+002
3.050e+002
[K]
0
1.500
3.000 (m)
0.750
2.250
Z
X

References

- **References**
- Analysis Of Steam Turbines - International Refereed Journal of Engineering and Science (IRJES) ISSN (Online) 2319-183X, (Print) 2319-1821 Volume 3, Issue 2 (February 2014), PP.32-48 by A.Sudheer Reddy MD, Imran Ahmed, T.Sharath Kumar, A.Vamshi Krishna Reddy, V.V Prathibha Bharathi.
- Introduction to CFD Basics by -Rajesh Bhaskaran and Lance Collins.
- CFD Meshing with ANSYS Workbench - March 14, 2013 by CAE Associates.
- **http://www.pbsenergo.com/products-and-services/turbines/steam-turbines/low-pressure-condensing-turbines.**
- Applied Computational Fluid Dynamics - André Bakker.
- Basics of computational fluid dynamics analysis - Chaitanya Vudutha, Parimal Nilangekar, Ravindranath Gouni, Satish Kumar Boppana, Albert Koether.
- http://www.pbsenergo.com/products-and-services/turbines/steam-turbines/low-pressure-condensing-turbines.
- Cfd and heat transfer blog.

Thank You!