Objective:

To simulate the Exhaust manifold in Ansys fluent for analysing the heat transfer and the flow of fluid inside.

Conjugate heat transfer:

The term Conjugate heat transfer is used to describe that when there is a heat transfer between solid to fluid and vice versa.

The conjugate heat transfer (CHT) analysis is an iterative way to simulate the heat transfer between solid and liquid surfaces.

Applications of CHT:

Heat exchanger, cooling fins with natural or forced convection, etc. . . .

K Omega model:

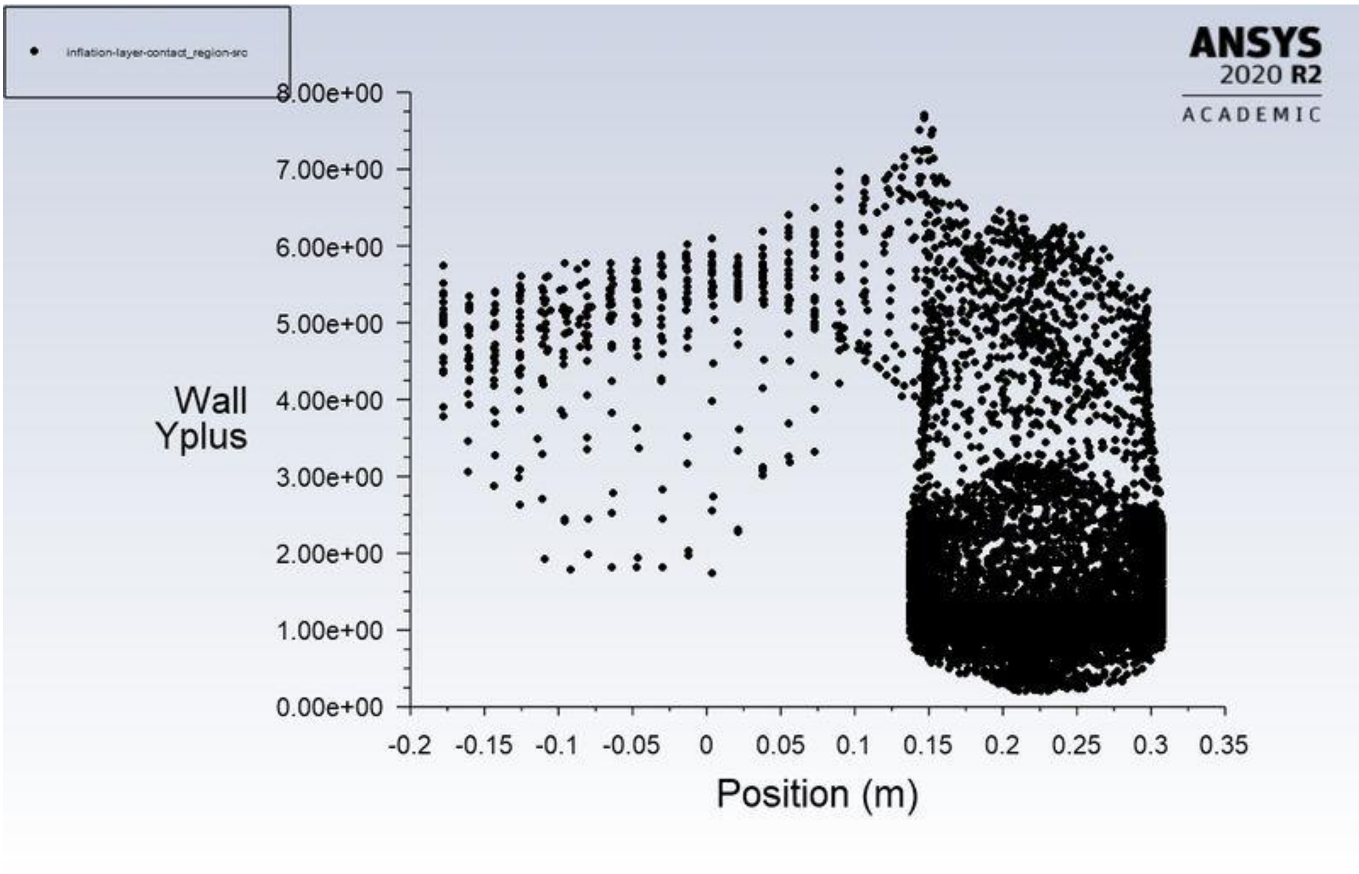
More accurate for near-wall interactions than k-epsilon models. Ability to predict flow separation and accurate prediction for internal flow, curvatures.

Needs small inflation layer which increases the total cell count.

For this model, Y+ should be below 10 (Y+<10).

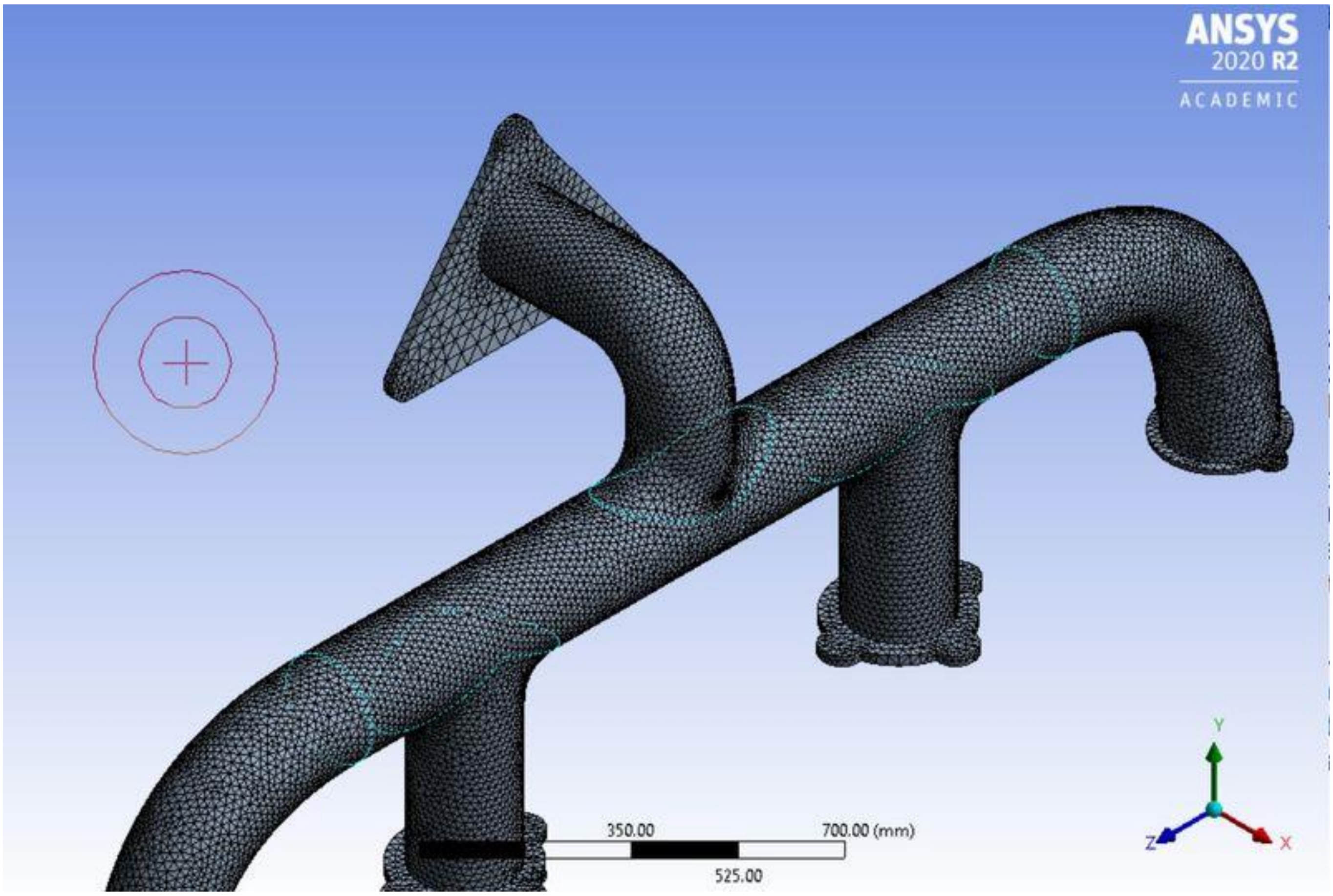
Using Matlab code total thickness is calculated with a desirable y+ of 3.

```
clear all
close all
clc

u = 5; %m/s
L = 0.15; %m
rho = 1.225; %kg/m3
mu = 0.000018375; %kg/m.s
Y_plus = input('enter desired value of Y+: ');

Re = u*L*rho/mu; %reynolds number

cf = 0.026/(Re^(1/7)); %skin friction coefficient

T_wall = 0.5*cf*rho*u^2; %wall shear stress

U_fri = sqrt(T_wall/rho); %friction velocity

del_s = Y_plus*mu/(U_fri*rho);

growthrate = 1.2; %growth rate of next cell from previous cell
no_layer = 6; % no of cells
tot_thick = del_s*2*1.2^(no_layer-1);

fprintf('n Reynolds number is %d n n',Re);
fprintf(' first wall thickness is %d in mn n',del_s);
fprintf(' Total thickness of the cell %d in mn', tot_thick);
```

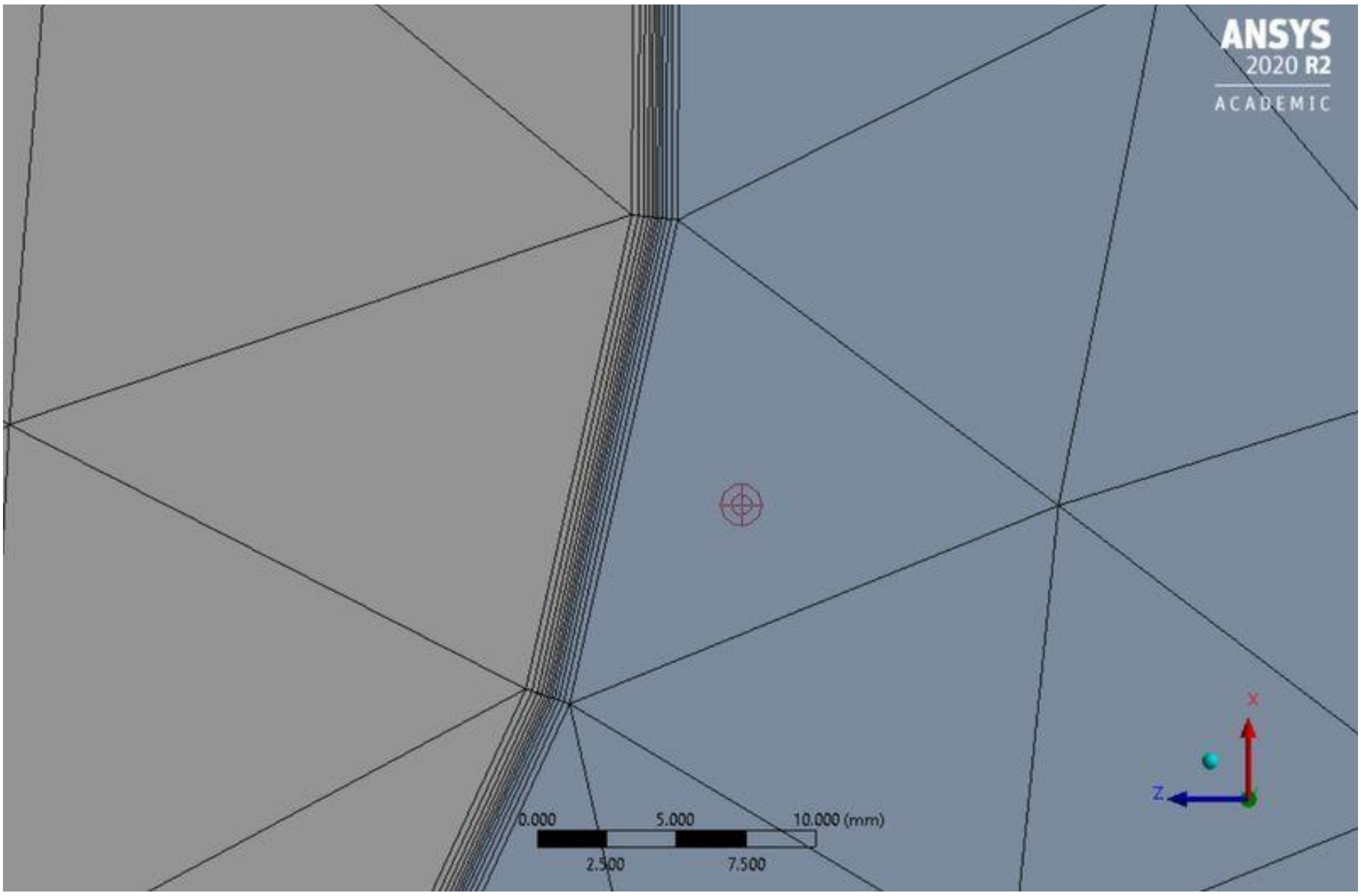


from the above chart, we can say that the y+ value is within 10. so we can use the K-omega model.

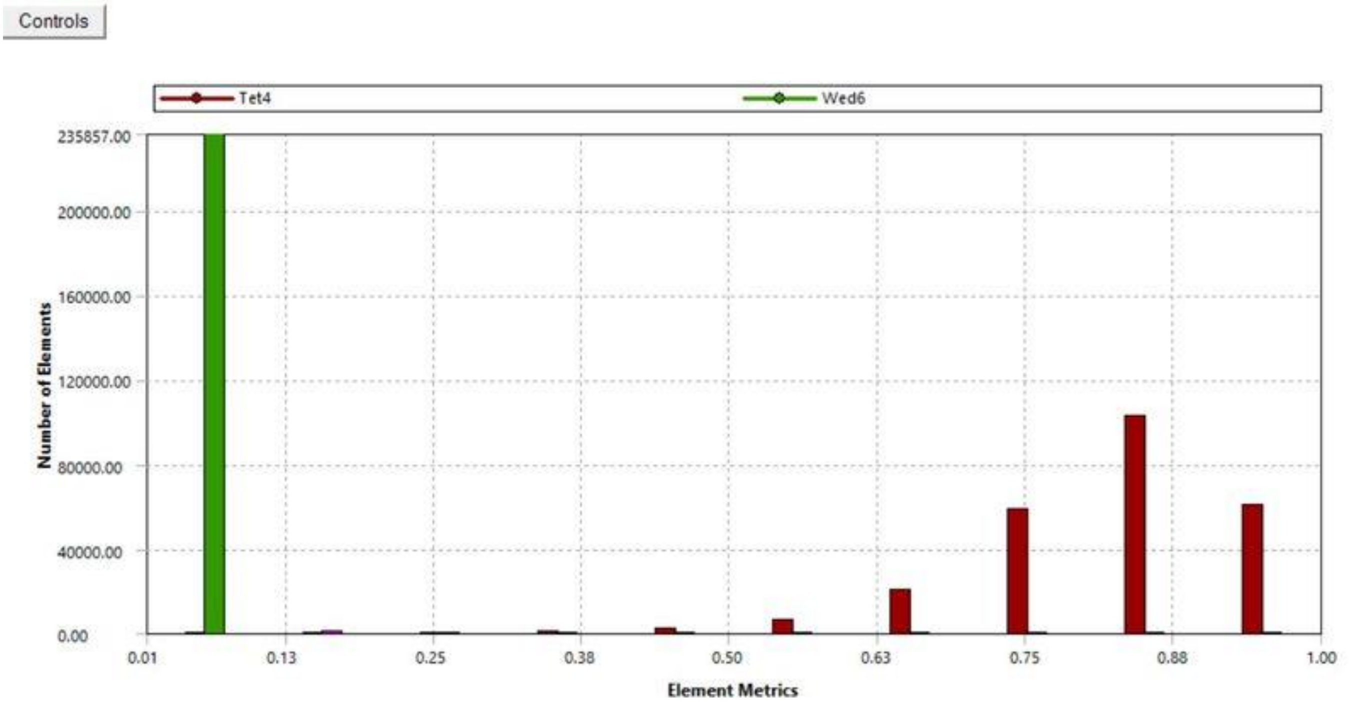
Meshing:



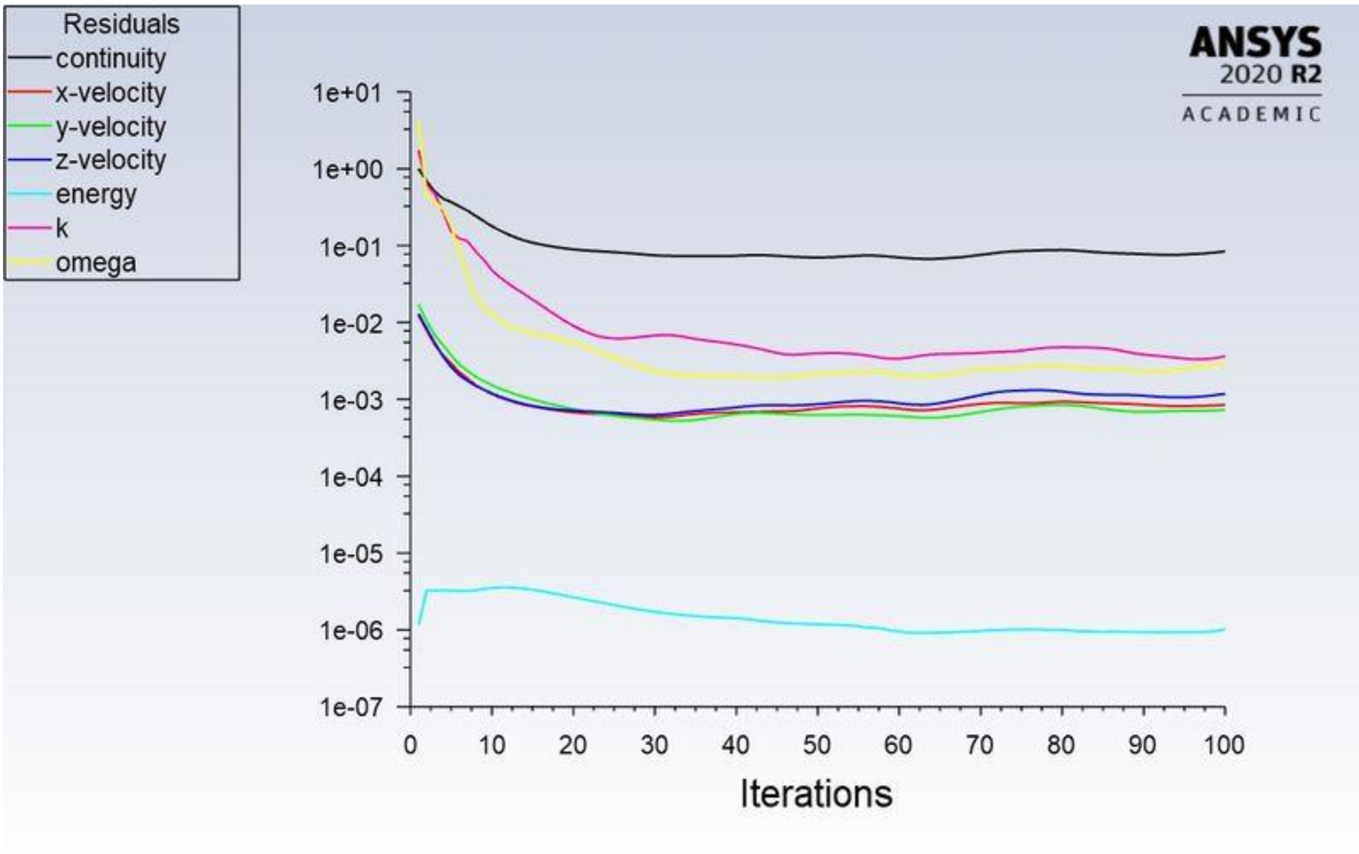
Inflation layer:



Element Quality:



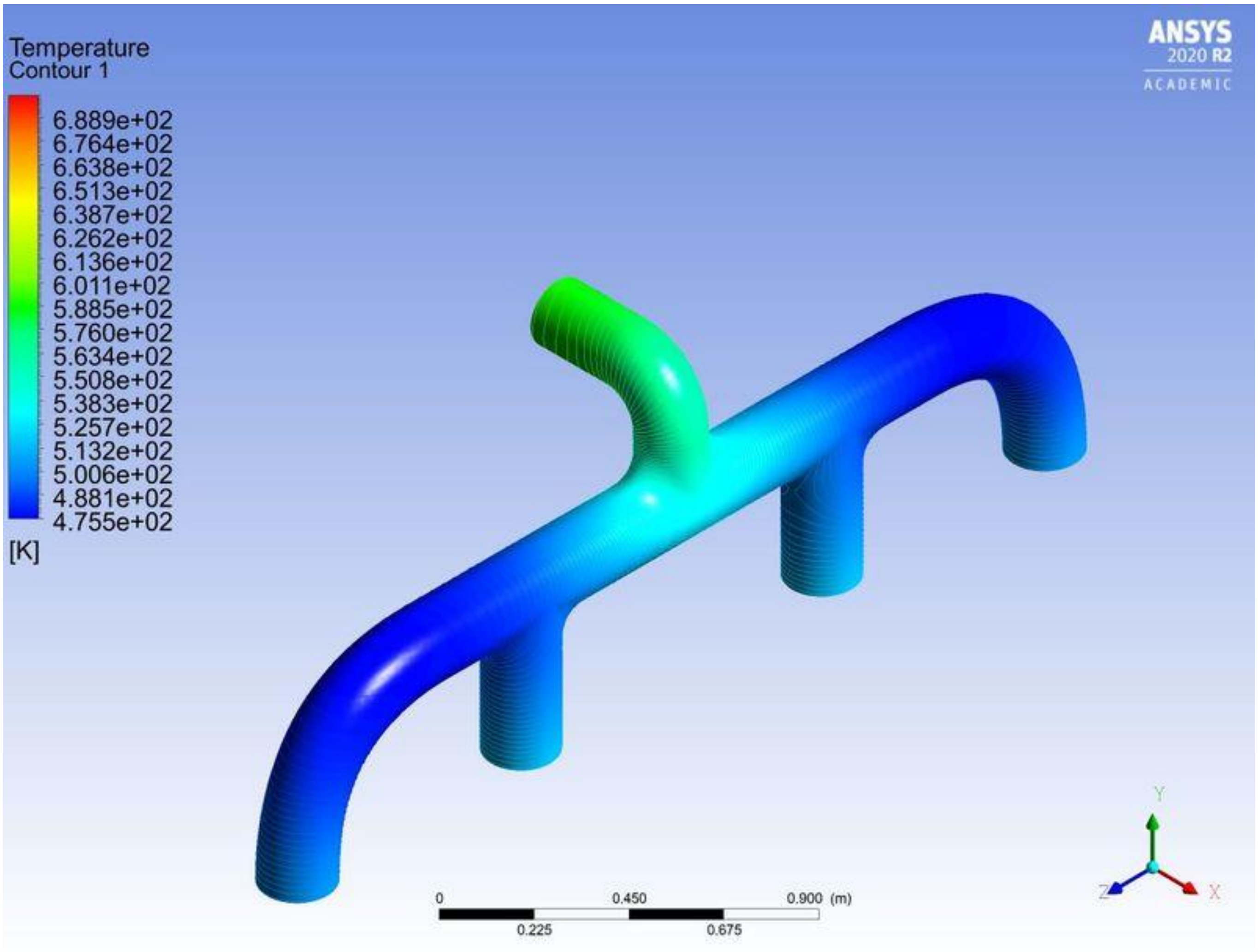
Residuals:



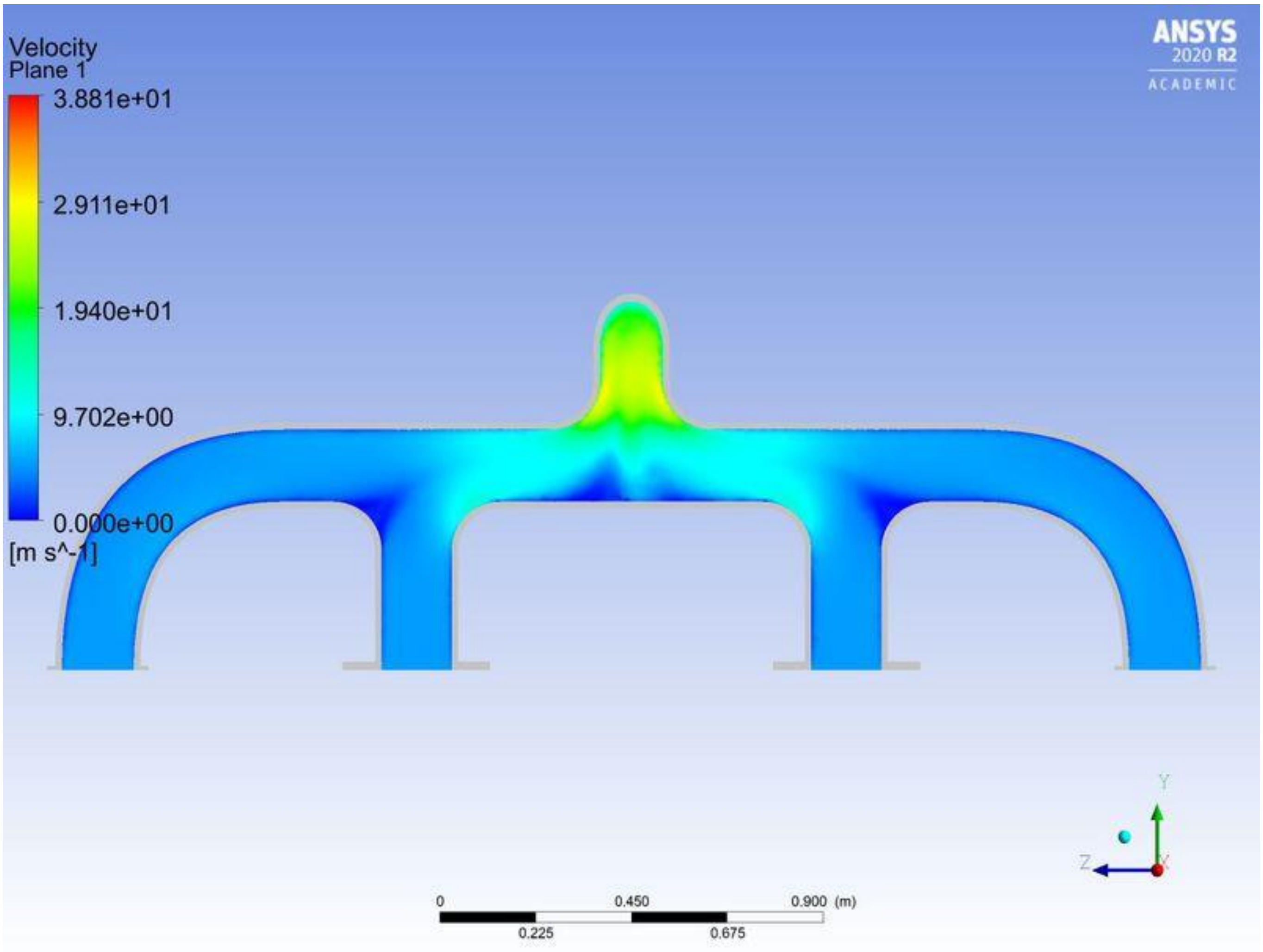
Since there is not much change in residual values and temperature variation in the temperature contour, thus the solution is converged.

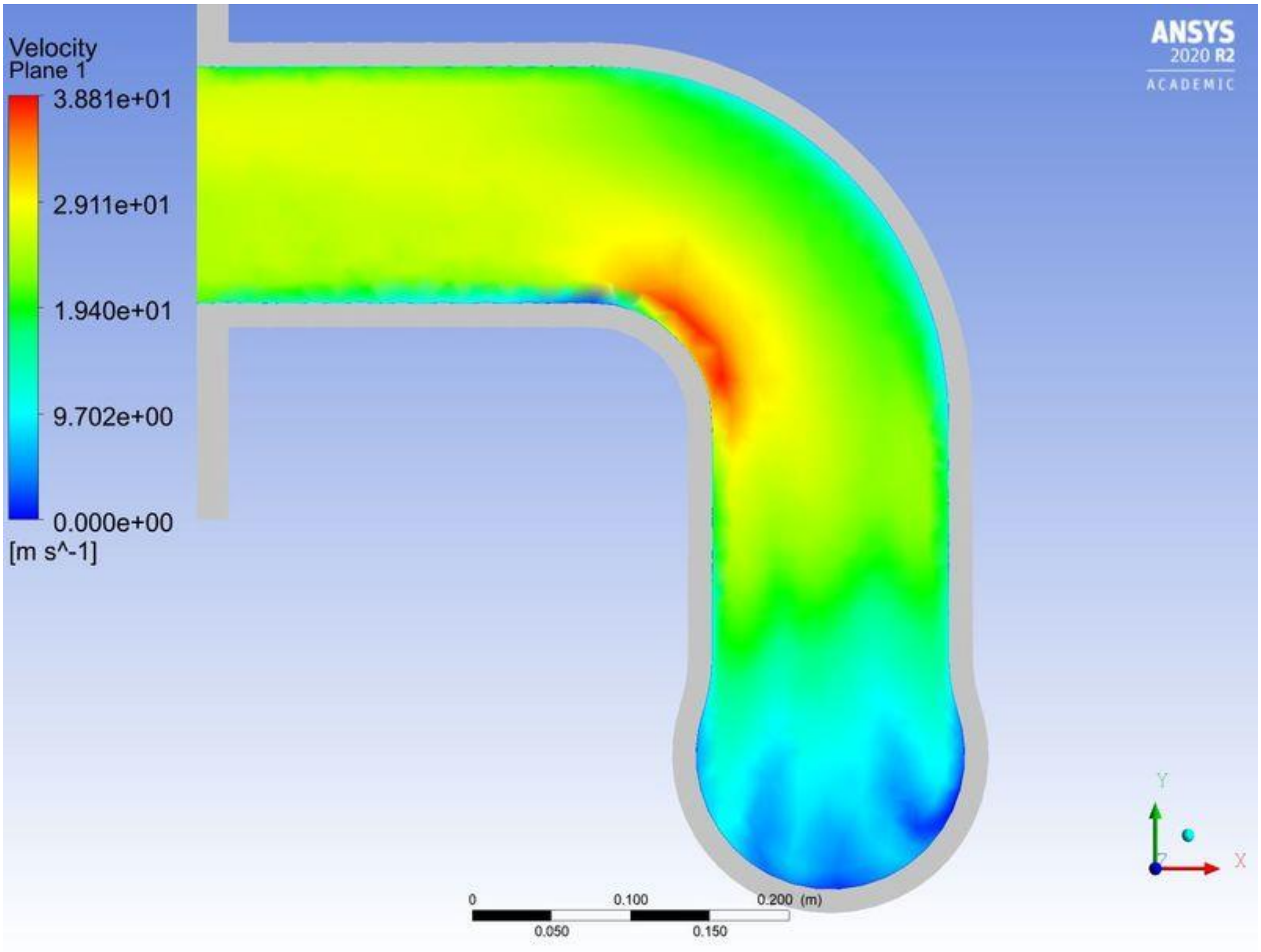
Post-processing:

Temperature contour:

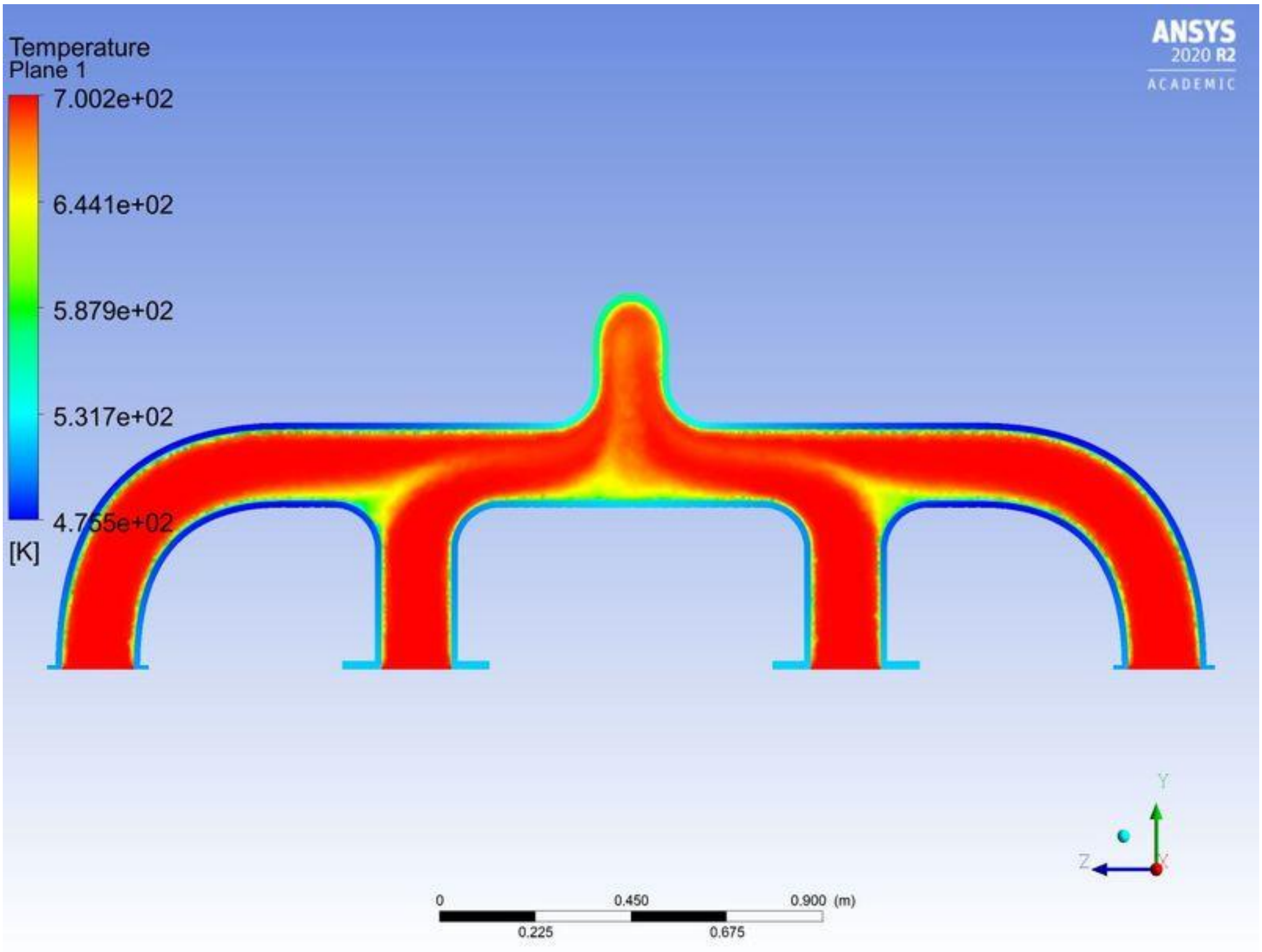


Velocity contour:

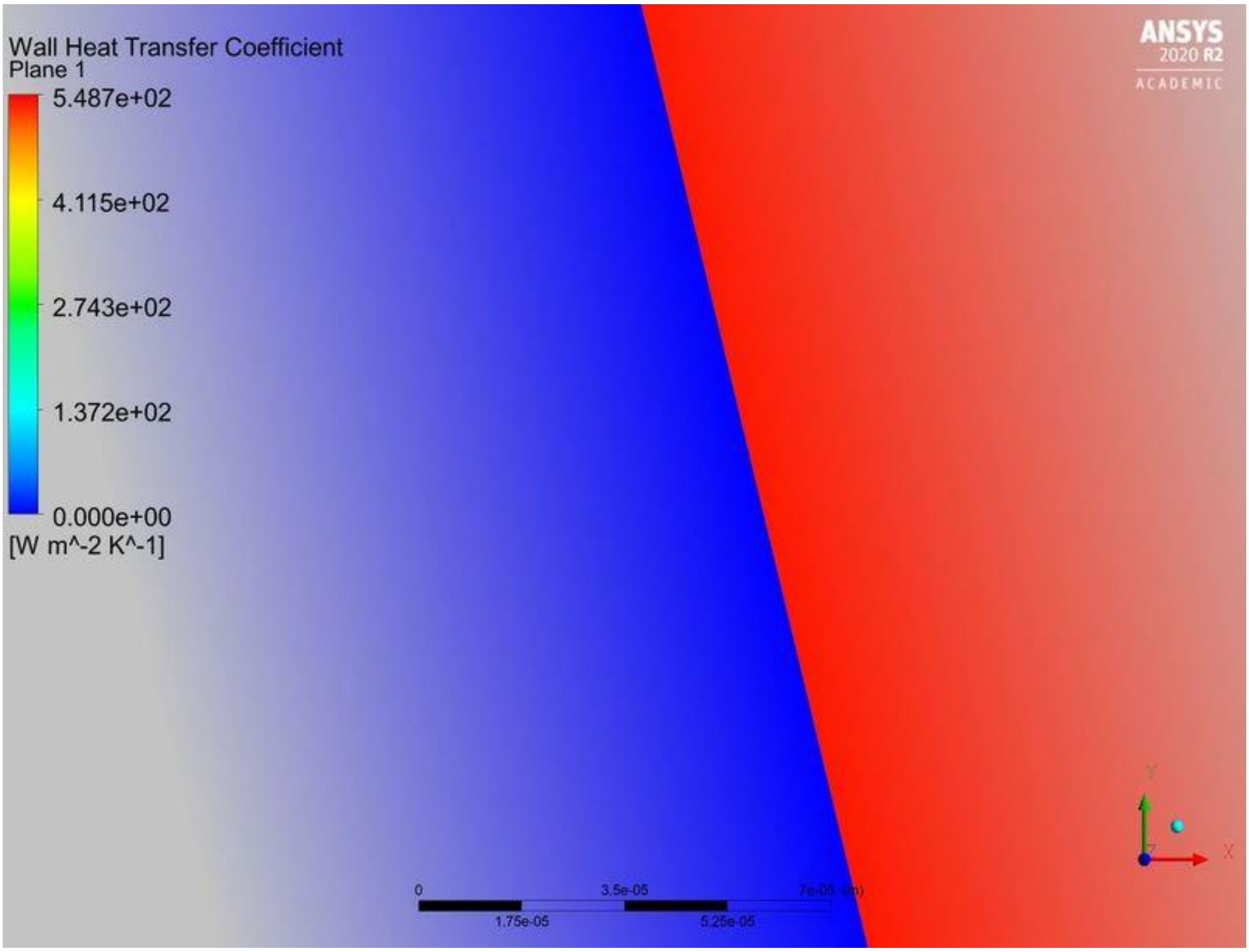



Velocity
Plane 1
3.881e+01
2.911e+01
1.940e+01
9.702e+00
0.000e+00
[m s^-1]
ANSYS
2020 R2
ACADEMIC
0
0.100
0.200 (m)
0.050
0.150
Y
X

Temperature contour:



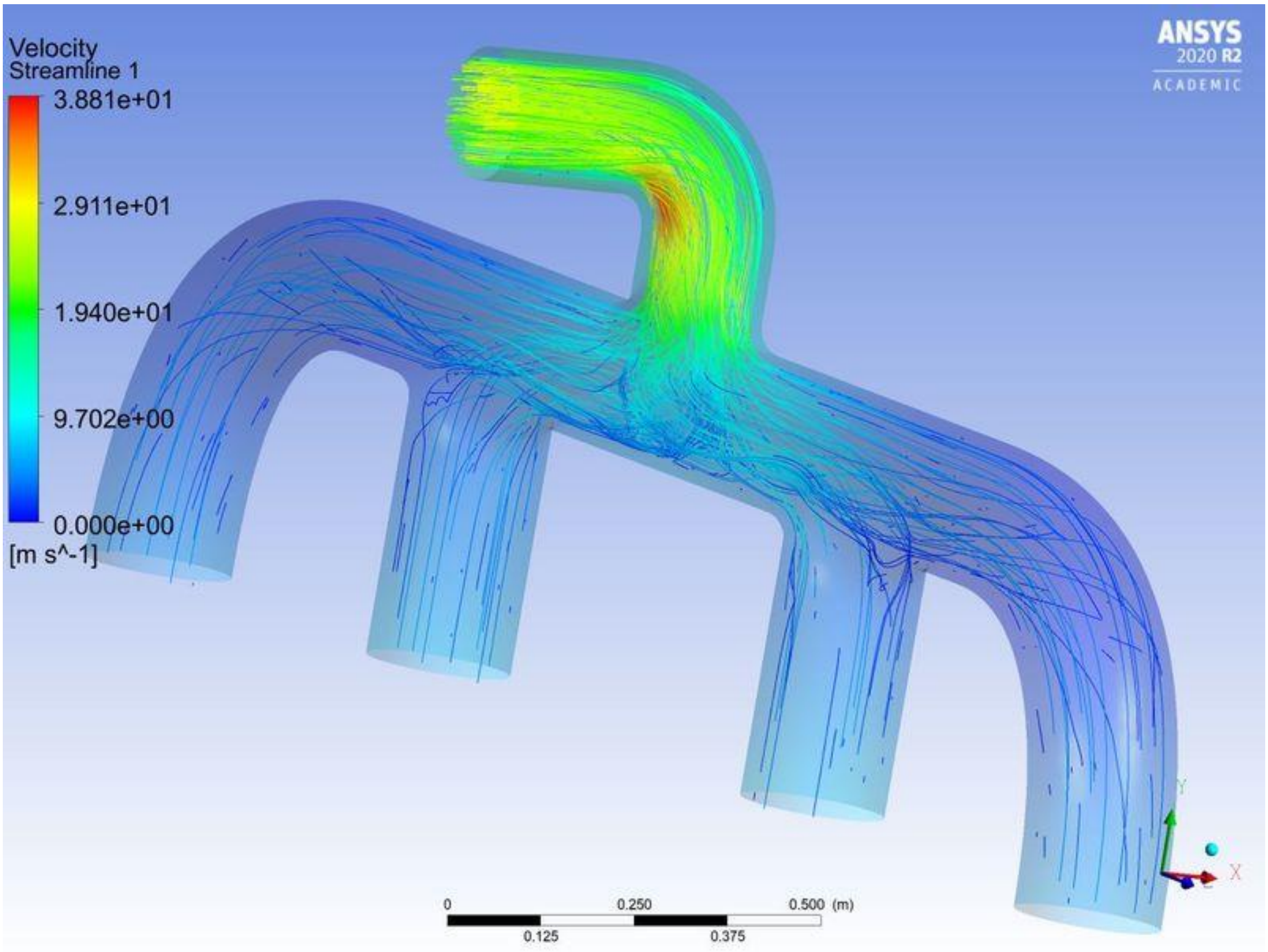
Wall heat transfer coefficient:



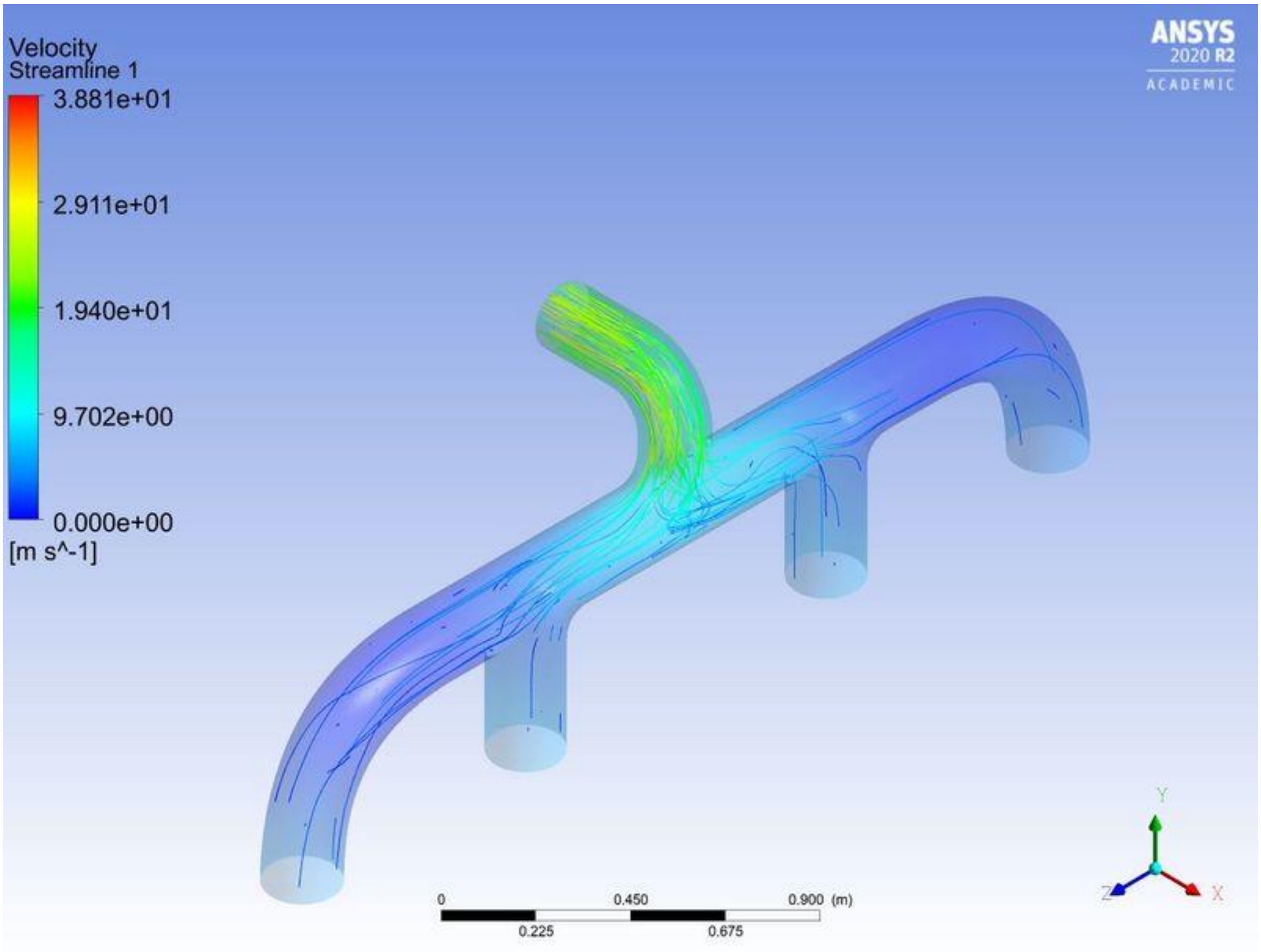
Area Average of Wall Heat Transfer Coefficient on inflation_layer

435.84 [W m^-2 K^-1]

Streamline:



Velocity
Streamline 1
3.881e+01
2.911e+01
1.940e+01
9.702e+00
0.000e+00
[m s^-1]
ANSYS
2020 R2
ACADEMIC
0
0.250
0.500 (m)
0.125
0.375
Y
X



To Verify the result:

Assuming the air as the fluid flowing in the pipe

For temperature t_air = 700k, from data book

Properties of air:

density = 0.524 `(kg)/m^3`

Kinematic viscosity = 63e-6`m^2/s`

Prandtl number = 0.678

Conductivity = 0.0521`W/m.K`

Reynold's number = `(u.L)/v`

= `(5*0.166)/(63.03e-6)`

Thus, *Re* = *`13.17e3`*

For internal flow Reynold's number > 10000

Nusselt number Nu = $0.036 \, Re^{0.8} \, Pr^{0.3}$

Thus, Nu = 63.31378

We know that, Nu = $\frac{h.D}{k}$

Thus which gives h = 19.8591 $W/m^2.K$

thus the h = 20 $W/m^2.K$ is used in the simulation

and it proves that the simulation is accurate.

(ii)as per the mass conservation law

we can say that the outlet velocity should be 4 times the inlet since there are 4 inlet and also the Inlet and outlet diameter are same.

Area Average of Velocity on outlet 25.2434 [$m \, s^{-1}$]

Thus the velocity is approximately 25 [$m \, s^{-1}$]

Conclusion:

Thus the simulation has successfully run in Ansys fluent.