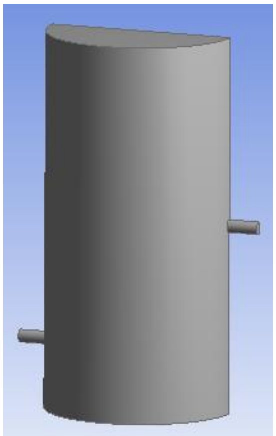
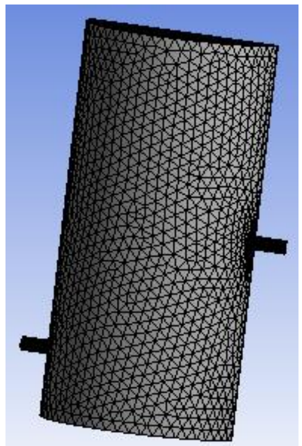
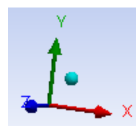
MAE 598: Project 1: Applied Computational Fluid Dynamics **VIKRAM SURESH**

Geometry and Mesh:

Base Dia: 0.6m
Inlet and Outlet diameter: 0.04m
Height: 1.2m
Extension of inlet and outlet pipes from tank lateral surface = 0.1m.
'z' represents height of the inlet and outlet from the bottom surface.
Mesh: Inflation given (5 layers)







Task 1: Temperature at outlet in 'K'

			Z1(m)	
		0.2	0.6	1
	0.2	304.5655602	304.4315528	301.8349951
Z2(m)	0.6	**305.74672**	303.1190986	301.4439722
	1	305.6110803	**303.6809143**	**301.4373442**

	Temp. (K)
Highest	305.74672
Median	303.6809143
Lowest	301.4373442

Method: Second order upwind for all equations.

Model: Turbulence k-eps Realizable model. (Except for task 4).

Steady flow, Pressure based

Fluid: Water with given properties.

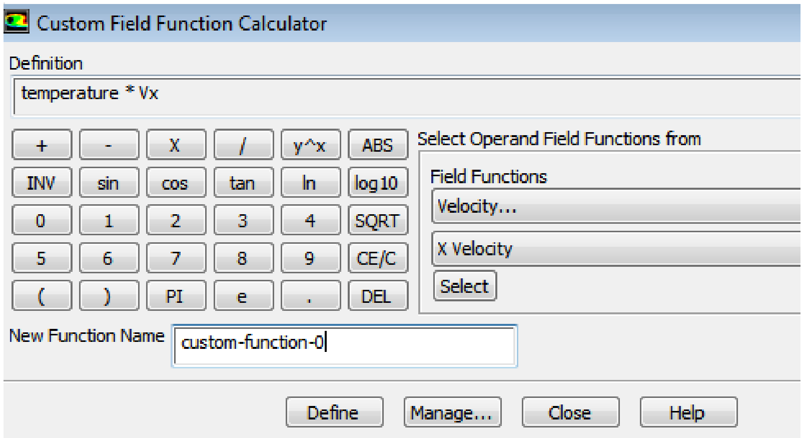
Base Temperature: 70 degree Celsius

Calculation of Outlet Temperature on Fluent. Part of Task 1

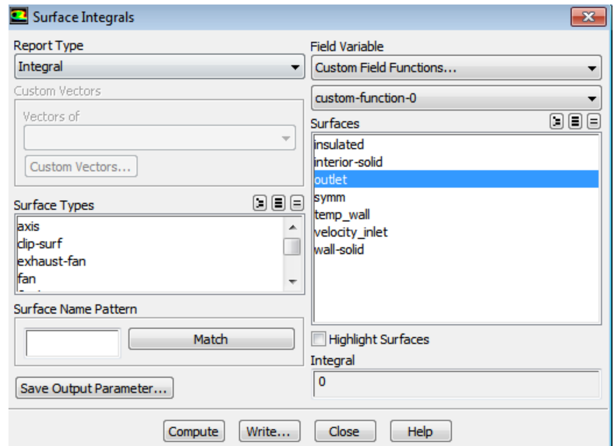
$$T_{out} = \frac{\int\int v_n \, T \, dA}{\int\int v_n \, dA}$$

Numerator of the equation:

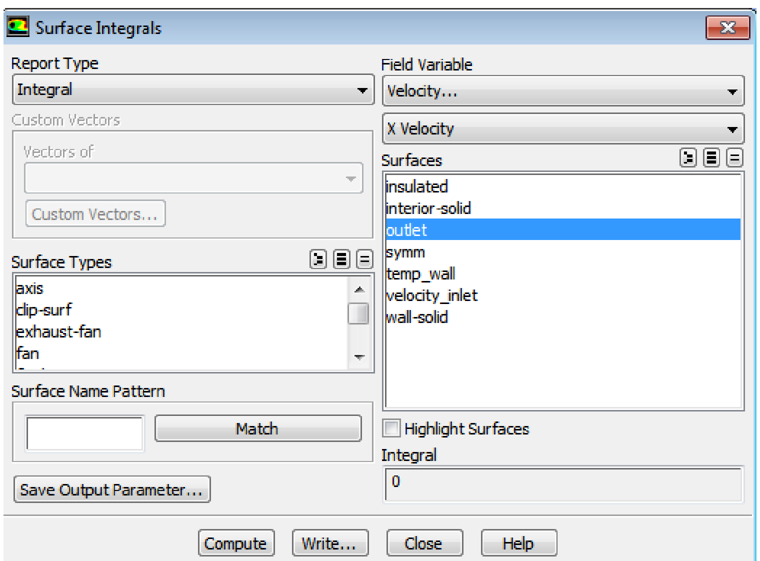
Defining custom field function



Surface Integrals under 'Reports'



Denominator of the equation:



The value obtained by dividing the numerator result and denominator result gives the required value of Temperature outlet. All the values in the table are obtained in the same procedure.

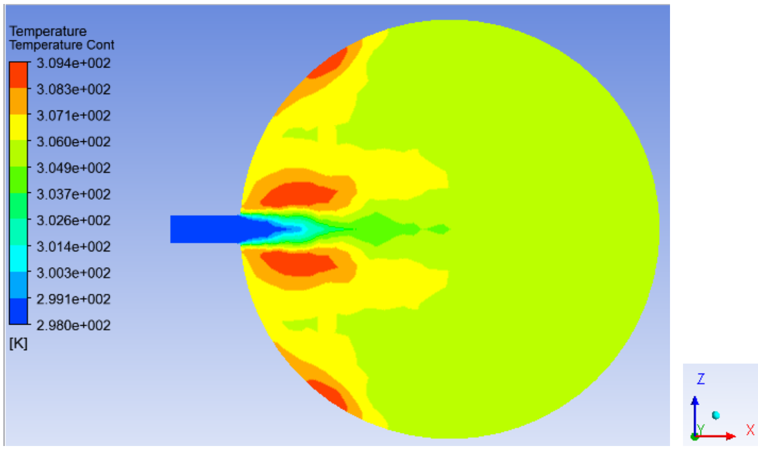
(Choosing 'mass-weighted average' gives the same result as with the above procedure probably because the density remains constant (cancels in numerator and denominator).

Report → Surface Integrals → Report Type: 'Mass-Weighted Average → Variable: Temperature)

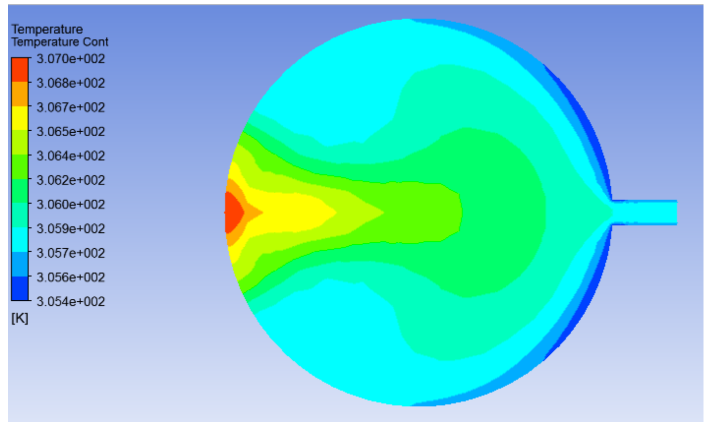
TASK 2:

Z1 = 0.2m, Z2 = 0.6m;   Highest Outlet Temperature Case

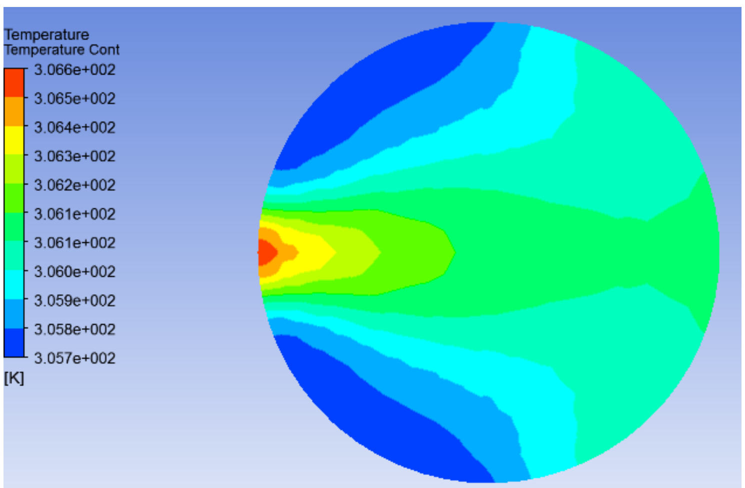
(i) <u>Cross-section at z = 0.2m</u>



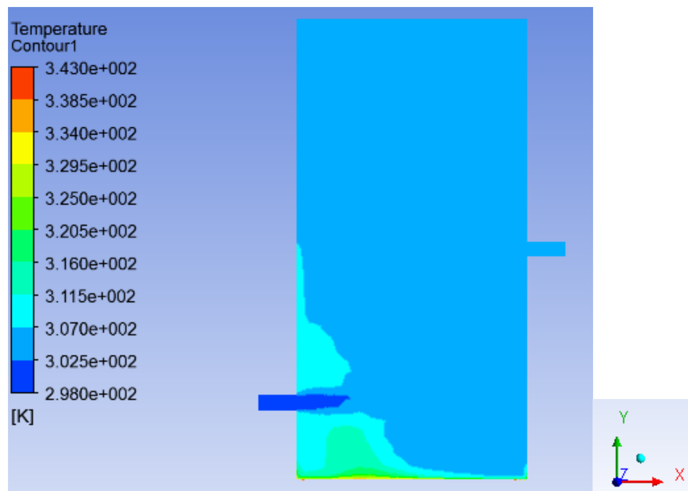
<u>Cross-section at z = 0.6m</u>



<u>Cross-section at z = 1m</u>

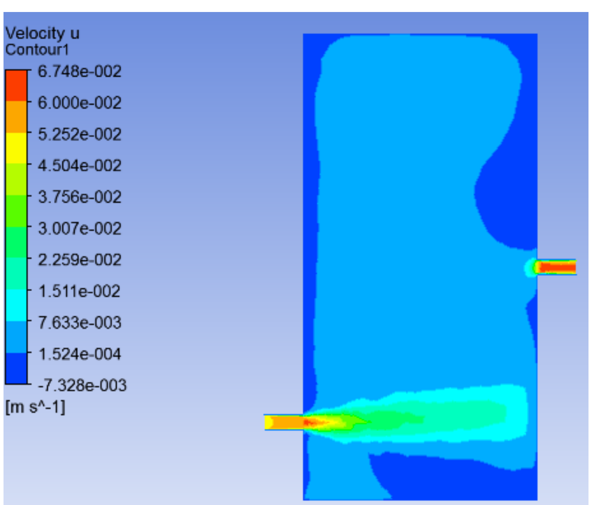


(ii) Temperature contour at symmetry plane

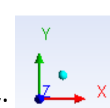


-

(iii) <u>u-Velocity contour at symmetry plane</u>

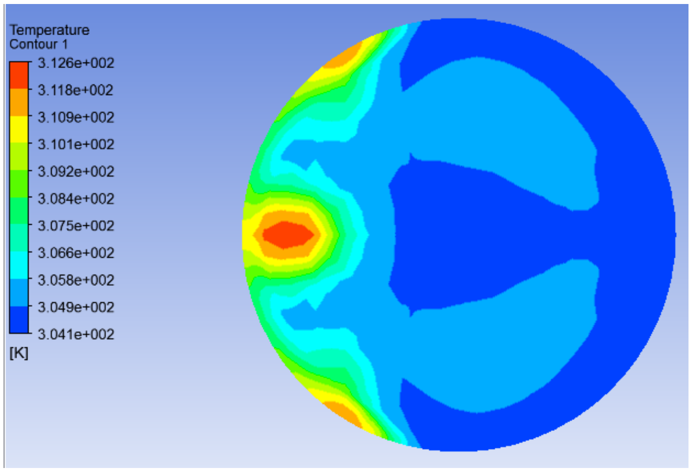


<u>← Plane of symmetry;</u>

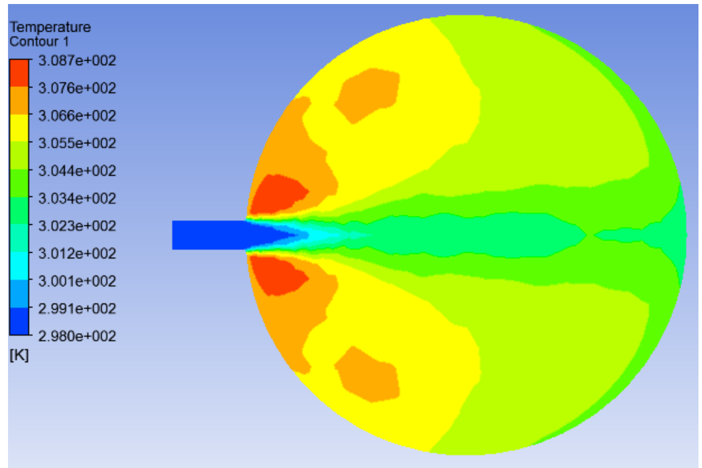


Z1 = 0.6m, Z2 = 1m;   Fifth highest outlet temperature case.

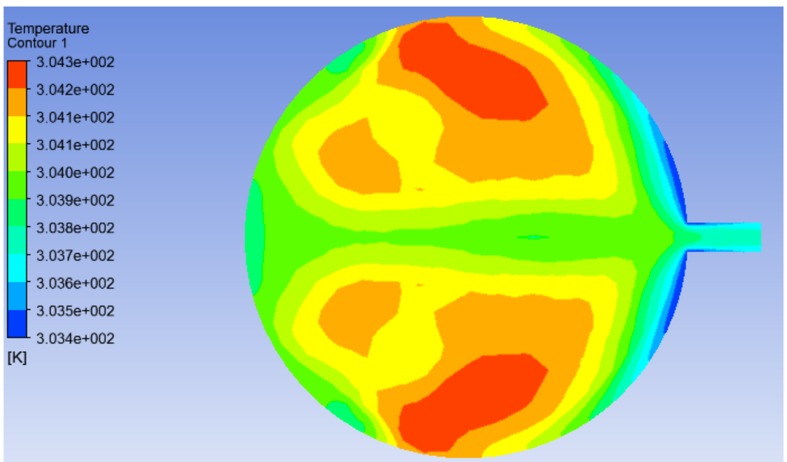
(i)   Cross-section at z = 0.2m



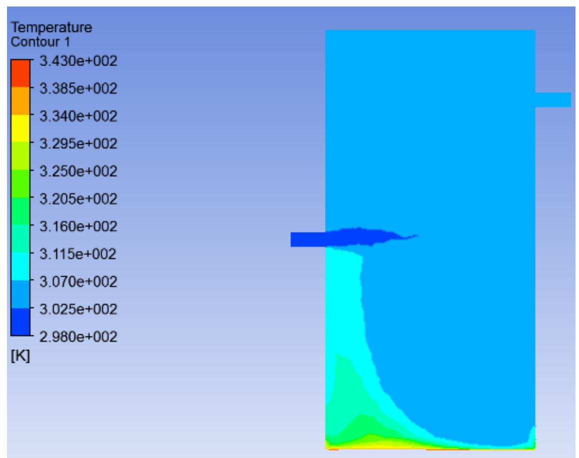
Cross-section at z = 0.6m



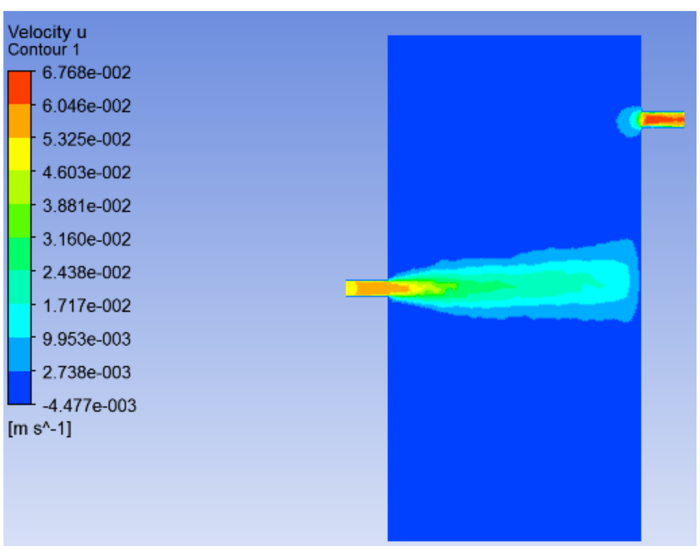
Cross-section at z = 1m



(ii) Temperature at symmetry plane

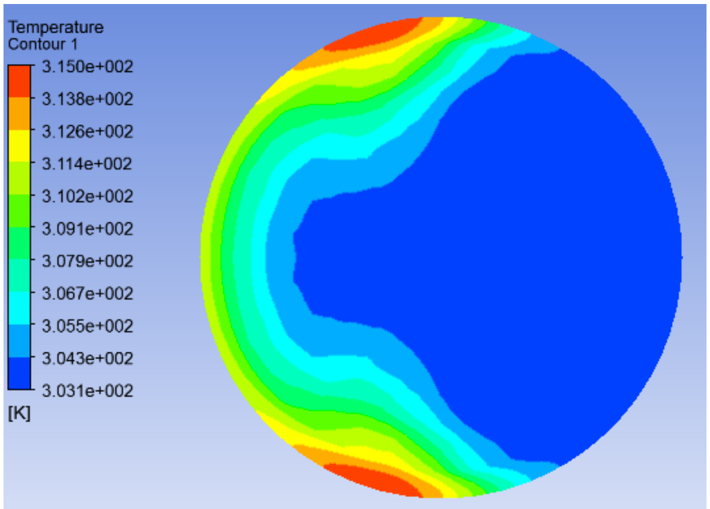


(iii) u-Velocity contour at symmetry plane

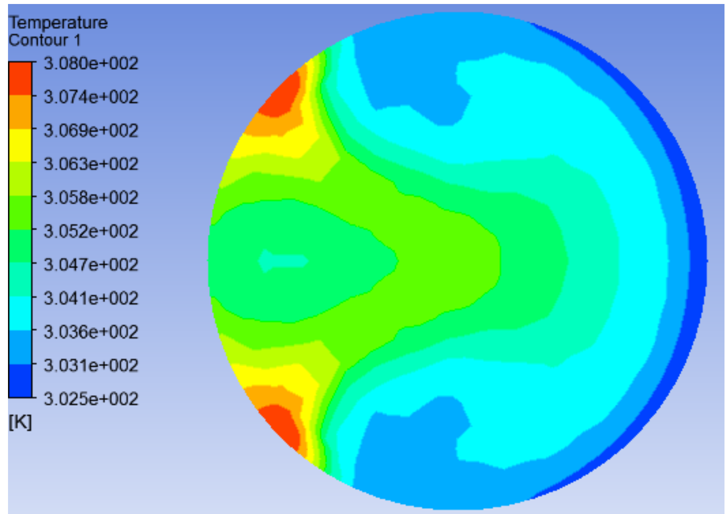


Z1 = 1m, Z2 = 1m; **Lowest Outlet Temperature case**

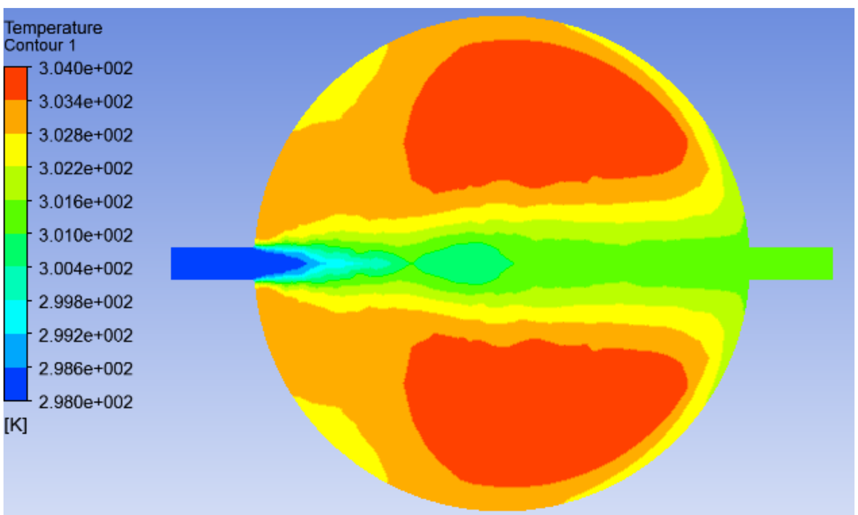
(i) Cross-section at z = 0.2m



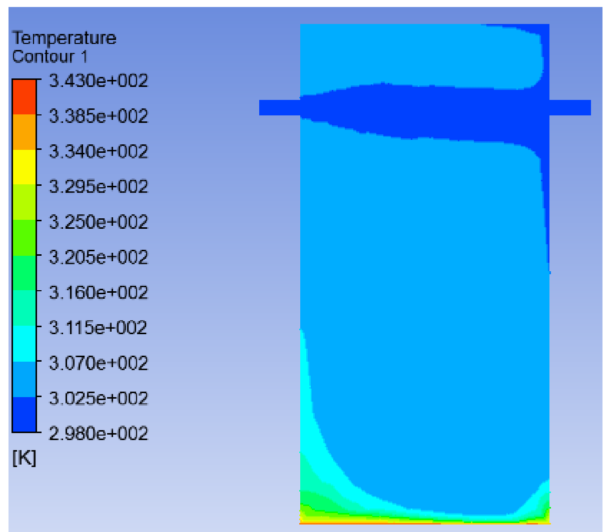
Cross-section at z = 0.6m



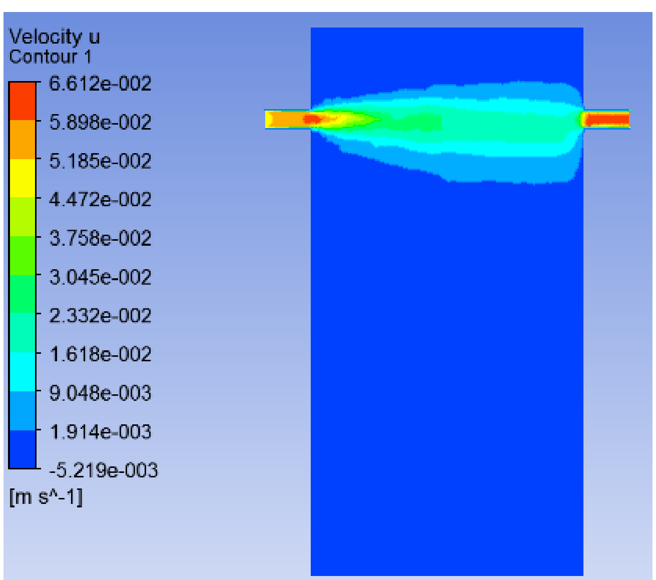
Cross-section at z = 1m



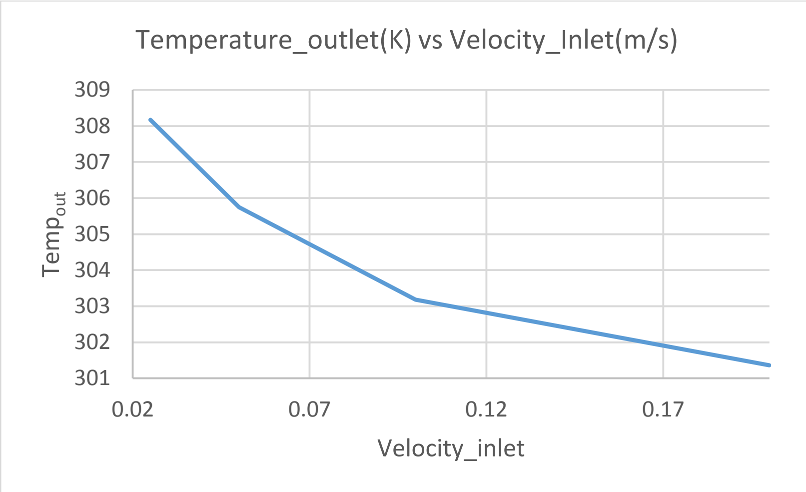
(ii) Temperature contour at symmetry plane



(ii) U-Velocity contour at symmetry plane

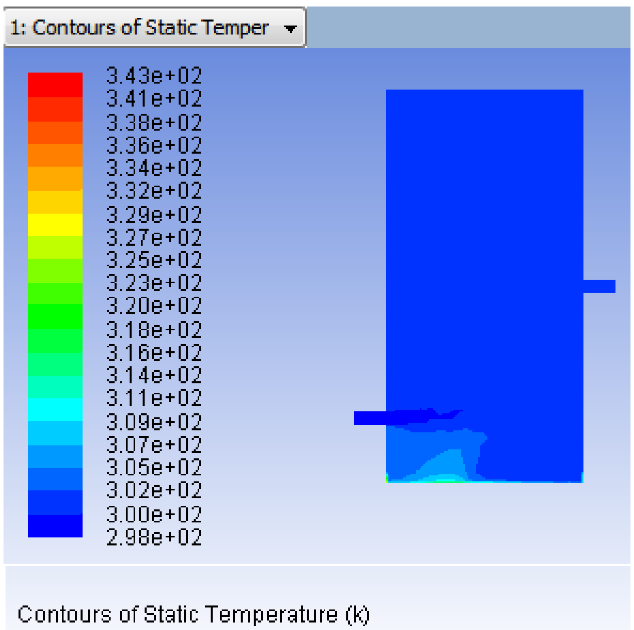


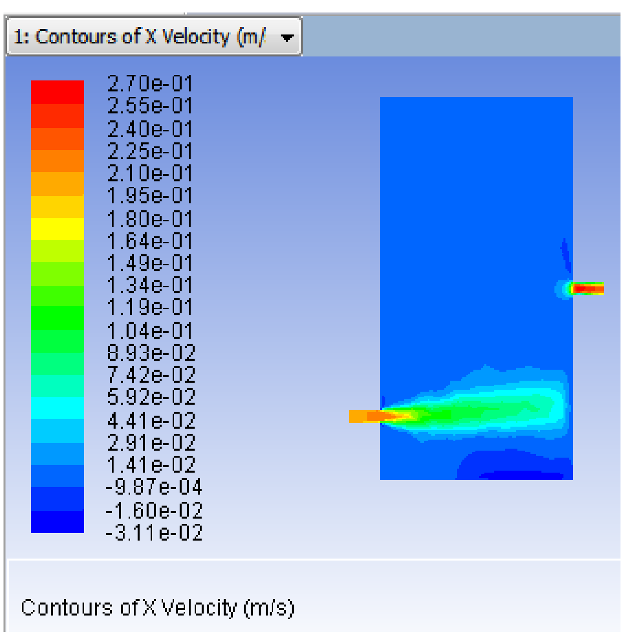
Task 3:



Inlet Velocity)m/s	Temperature outlet(K)
0.025	308.16656
0.05	305.74672
0.1	303.17868
0.2	301.36284

0.2m/s: z1=0.2m, z2 = 0.6m





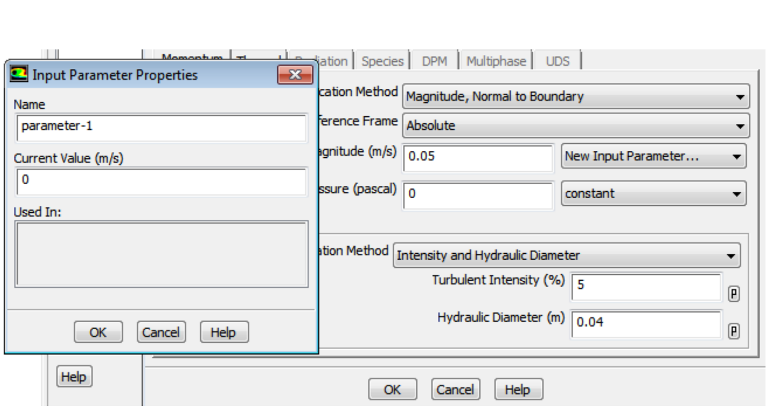
‘Local scale’ was used for the cross sections z=0.2m, 0.6m, 1m. Not ‘Global Scale’.

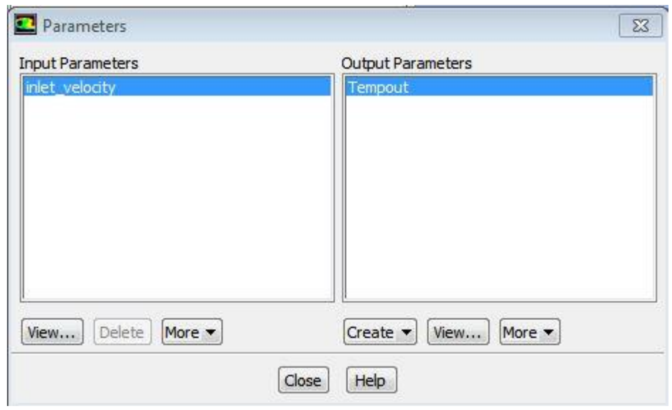
In **task 2**, from the temperature contour in symmetry plane for Z1=1m and Z2=1m, we observe that the gradient in temperature is only near the bottom surface. So, for this case, we do not expect much increase in outlet temperature. We can also say that the fluid has less time to raise to a high temperature since Z2=1m.

For the highest case: When Z1=0.2m, the inlet is close to the surface where the gradient in temperature is more. As it moves to the outlet, Z2=0.6m, the temperature raises till the highest value mentioned in the table.

Task 3 was run using ‘Parametric design’ approach.

Select ‘New Input Parameter’ from velocity inlet Magnitude as shown → Name the parameter (‘inlet_velocity’ here) → enter the value of velocity (the current value) → Use ‘Create’ icon to create an output parameter (Here ‘Static Temperature’ at outlet). → We can now see a 7th row named ‘Parameter’ (in the current Workbench Project) as shown in the 3rd figure below).





Duplicating ‘Design Points’: Right click on ‘DP 0’ and duplicate. Enter the required value of velocity in each duplicate and then click ’Update Selected Design Points’. Now, all the 4 cases will run one after the other on its own and then display the value of outlet temperature in a new column. We can then right click and export each ‘DP’ individually and open as a new project and edit further, if required.

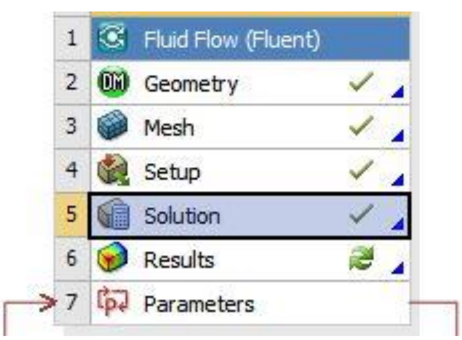


Table of Design Points

	A	B
1	Name	P1 - inlet_velocity
2	Units	m s^-1
3	DP 0 (Current)	0.05
4	DP 1	0.025
5	DP 2	1
6	DP 3	0.2
*		

Task 4:

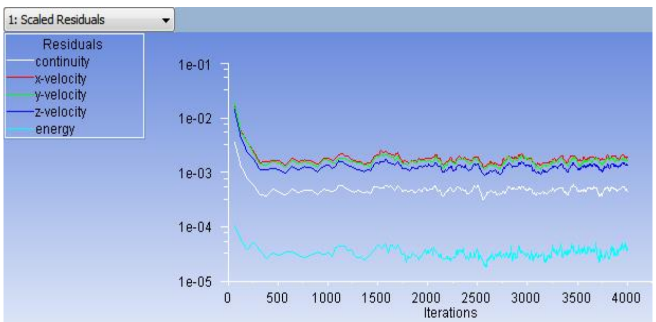
Largest outlet temperature value z1=0.2, z2=0.6 case with Laminar Model

Temperature at outlet: **303.9146 K**

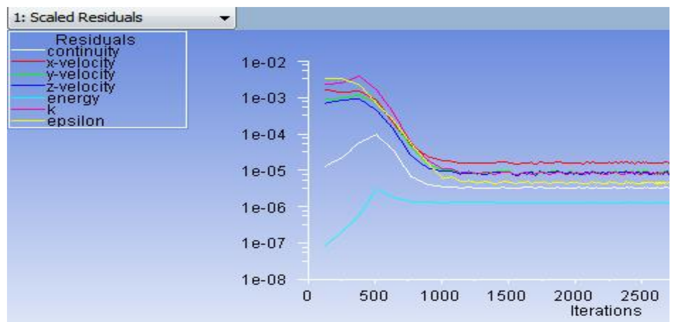
Running the laminar model on this turbulent flow gave an outlet temperature that is not very different from the one obtained for turbulent case. For this problem statement, the outlet temperature value is not much sensitive to the change of model. For a different problem, this might not be true. By switching to laminar model, we will not be able to capture the turbulence effects in the important areas in the domain.

When compared to the 'Turbulence model' case, the laminar model had more fluctuations in the residual curve. Owing to these fluctuations even after many iterations, it would be reasonable to take the average of the last 1000 iterations values. Further, the residual I energy curve doesn't go till e^-5 in the laminar case, even after 4000 iterations, whereas when modelled in the turbulence case, it does.

Laminar Model:



Turbulence Model:



Task 5:

a) Heat Transfer Rate at the bottom surface chosen from 'Reports'→ 'Fluxes' = ***1009.2791 Watts***
Area = Half of the base area since the geometry is symmetric.
   ⇨ Area = (pi/8)*(d^2) = (pi/8)*0.6*0.6 = 0.14137167 m^2
   ⇨ Heat Flux from bottom surface = (1009.2791/Area) = **7139.18923 W/ m^2**

b) Temperature at outlet after replacing the boundary condition from 'Temperature' to 'Heat Flux' with the above obtained value: **305.72688** K.
This value is very close to what was observed previously. Not much difference.