Swapnil Nimse   **Project – 1 Challenge #2**

Project Overview:
Using Ansys-Fluent, analyze dependency of the steady-state temperature at different parts of the system on the flow velocity at the inlet and buoyancy-driven thermal convection using the parametric design tool.
Process:
All the setup is used from the task 1A to perform this analysis unless otherwise stated. To analyze the various inlet flow velocity conditions, the parametric design tool is used. For defining parameter of velocity_inlet Ansys_fluent_setup is used. For defining parameter of T_out and T_mid CFD-Post is used. The steps which are modified from task 1A to run this case, are shown below
Boundary Conditions-
At Velocity inlet: velocity = 0.05m/s, temperature = 298.15 K,
                    In specification method: turbulence intensity = 5%, Hydraulic diameter = 0.04m
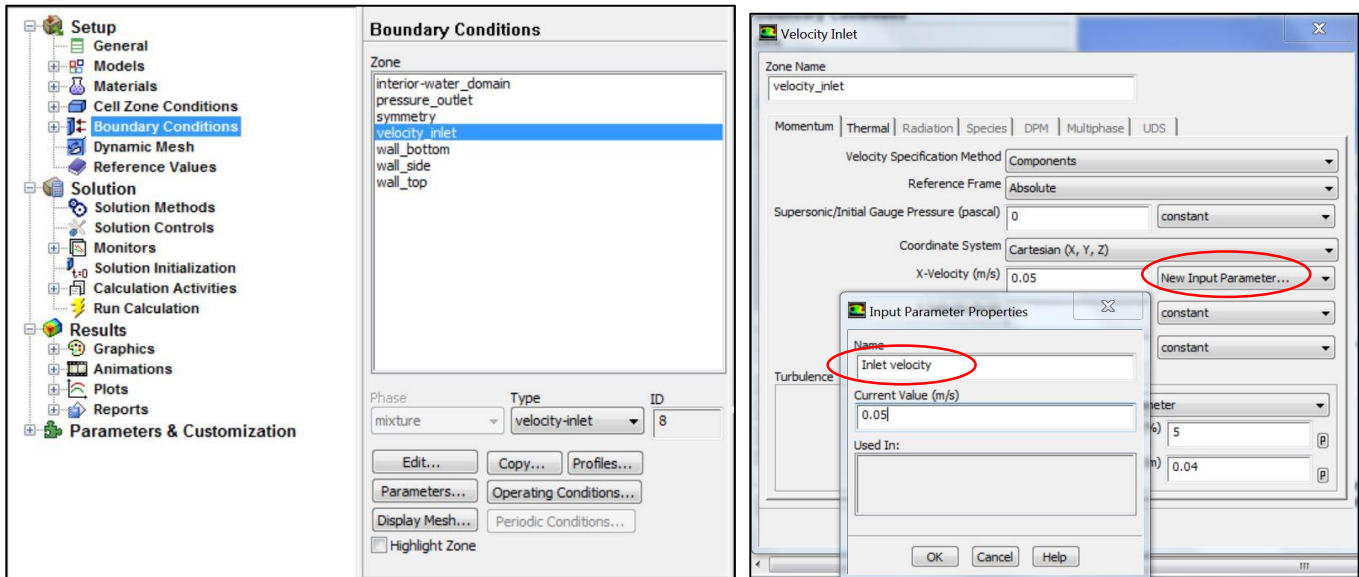Here the new parameter is selected for velocity as shown in figure 1.

Figure 1 : Window to setup new parameter for parametric design

Follow the steps as: Boundary Conditions > velocity_inlet>New Input parameter>Name (Inlet velocity) and current value (0.05) > ok
With keeping rest settings for other zones as it is.
Solution Methods-
Used pressure-velocity coupling with simple scheme using second order discretization.
Keeping other options as it is and initializing the solution from velocity inlet, the calculation is run for initial 500 iterations.
Solution got converged after 454 iterations. After convergence the case is accessed using CFD post for calculating further results.
To find the average temperature at outlet face and mid plane following function is used.

$$\text{T}_{\text{out}} = \frac{\iint v_n \, T \, dA}{\iint v_n \, dA}, \tag{3}$$
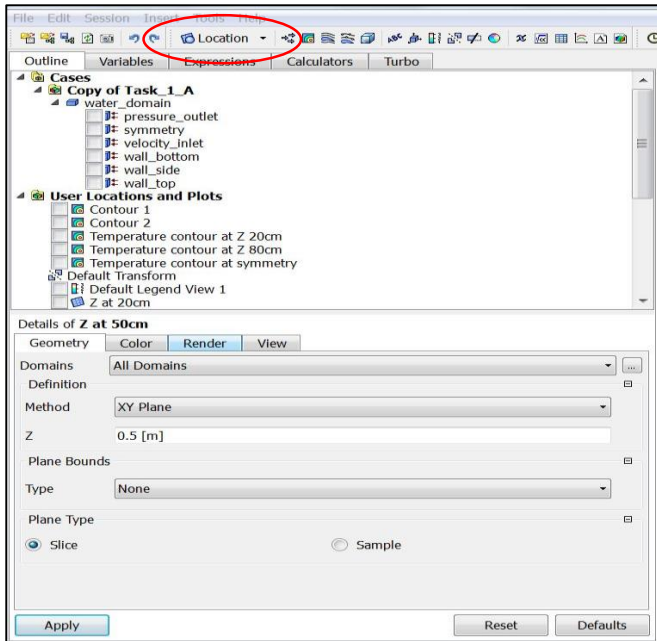
Where,

v_n : The non-uniform velocity normal to the outlet
T   : The non-uniform temperature at the outlet

using 'Expression' option in CFD Post, two functions are defined one for $\iint v_n \, T \, dA$ & other for $\iint v_n \, dA$. First the mid plane is created at Z = 0.5m as shown in figure 2.
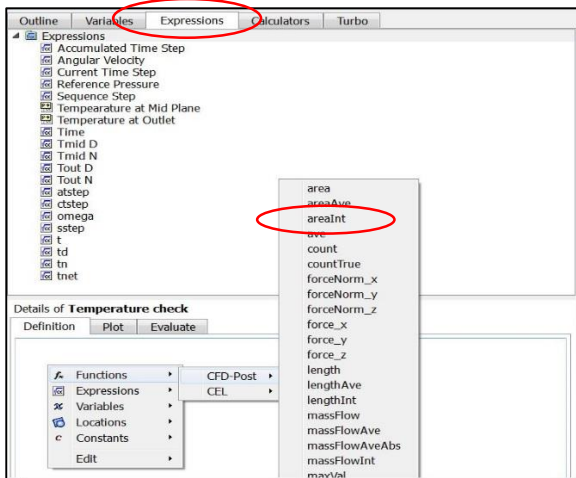
Figure 2 : Window to define new plane

After generating plane click on expression to define function for calculating temperature at outlet face and mid plane using areaInt option. Follow the steps shown in figure 3.

Location > plane > give name (Z at 50 cm)> apply

Figure 3 : window to define new expression

Expression > New expression > give name > right click in details window > function > CFD-Post > areaInt

In this case function name is Tout N:

areaInt > right click > variable (here temperature and velocity)>right click > location (here pressure_outlet)
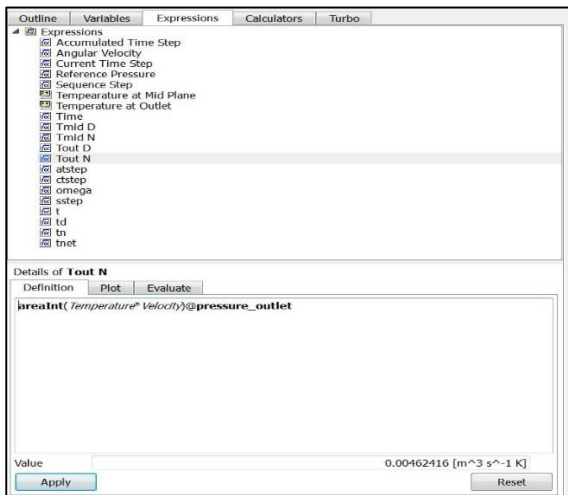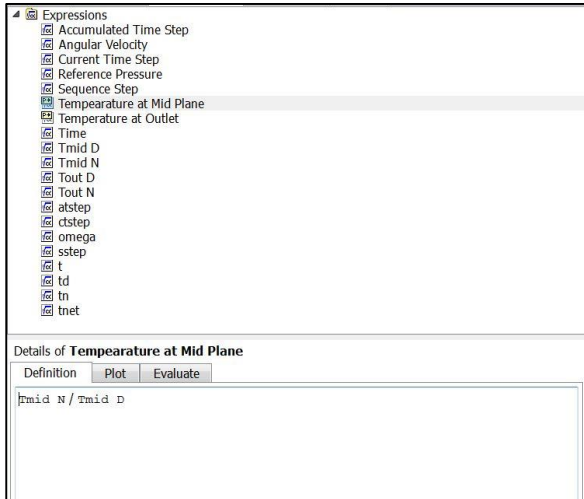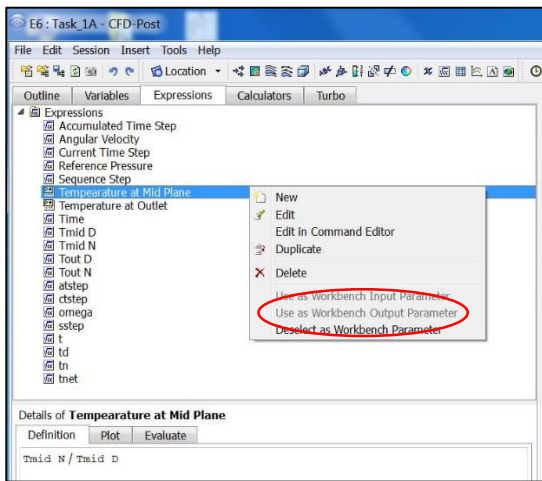
Define other expressions similarly

Figure 4 : window to define area integral

Calculating net temperature at outlet face and mid plane

New expression (here Temperature at Mid plane) > Tmid N / Tmid D

Similarly define for outlet face

Figure 5 : Window to define resultant temperature

Then to make the required expression as parameter

Right click on required function > use as workbench output parameter

Figure 6 : Window to defind function as output parameter

Close the CFD post.
You will see the following window in main Ansys display shown in figure 7. Click on parameter set
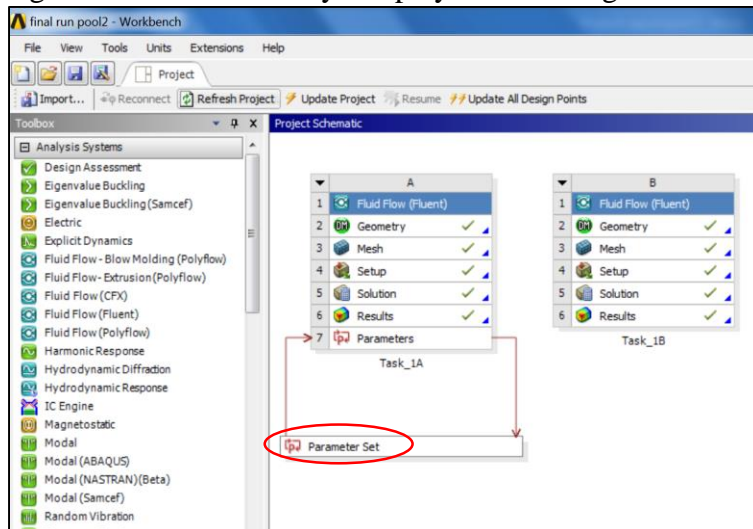

Figure 7 : window for initial parametric design

You will see the DP 0 (current) as 0.05. Define all other required value. In this case from 0.01 to 0.1 as shown below. Click on retain all points and then update all design points.
(Note after this it will take a while to compute all the design points and we can't stop solution in between)
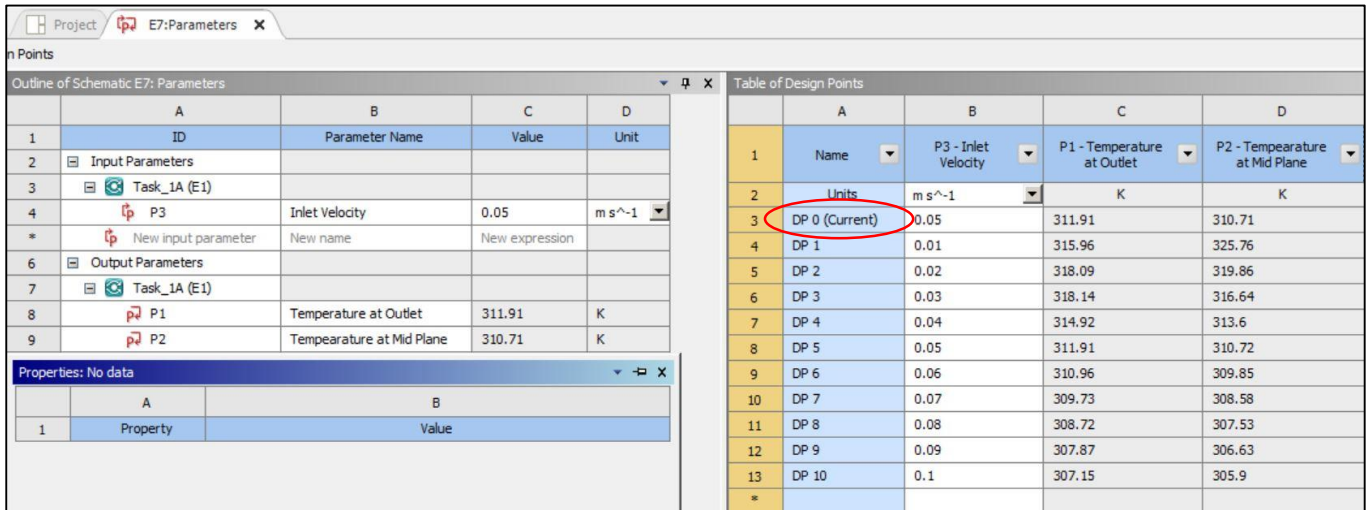
	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	⊟ Input Parameters			
3	⊟ 🟢 Task_1A (E1)			
4	🔷 P3	Inlet Velocity	0.05	m s^-1 ▼
*	🔷 New input parameter	New name	New expression	
6	⊟ Output Parameters			
7	⊟ 🟢 Task_1A (E1)			
8	📄 P1	Temperature at Outlet	311.91	K
9	📄 P2	Tempearature at Mid Plane	310.71	K

Properties: No data

	A	B
	Property	Value
1		

Table of Design Points

	A	B	C	D
1	Name ▼	P3 - Inlet Velocity ▼	P1 - Temperature at Outlet ▼	P2 - Tempearature at Mid Plane ▼
2	Units	m s^-1 ▼	K	K
3	DP 0 (Current)	0.05	311.91	310.71
4	DP 1	0.01	315.96	325.76
5	DP 2	0.02	318.09	319.86
6	DP 3	0.03	318.14	316.64
7	DP 4	0.04	314.92	313.6
8	DP 5	0.05	311.91	310.72
9	DP 6	0.06	310.96	309.85
10	DP 7	0.07	309.73	308.58
11	DP 8	0.08	308.72	307.53
12	DP 9	0.09	307.87	306.63
13	DP 10	0.1	307.15	305.9
*				

Figure 8 : Window for how to set parametric design

After completion of the run we get the t_out and t_mid for all the given values of inlet velocities.
Extract the data into excel or Matlab to draw the line plots of t_out and t_mid w.r.t. inlet velocity. Here I have used Matlab.
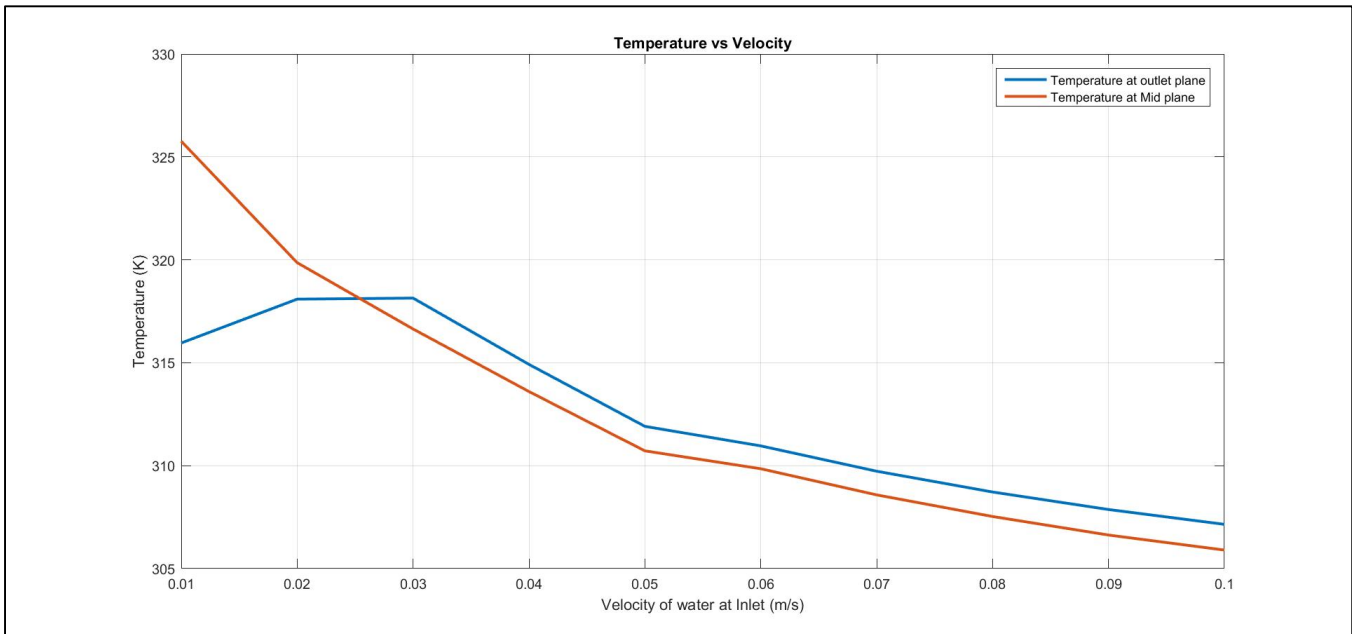

Figure 9 : Line plot of temperature vs inlet velocity at outlet and mid-plane

From the above graph it is clear that as we increase the velocity at inlet, the temperature at outlet and mid_plan is decreasing. But, in this case at low velocity values (0.01 and 0.02), I am getting the reversed flow at outlet. Hence the values obtained for temperature at outlet at V = 0.01 and 0.02 are incorrect. If we perform the same analysis with boundary condition type at outlet face as 'outflow' instead of 'pressure_outlet' then issue of reversed flow is not there. We can also refine the mesh to avoid the reversed flow. But due to the limitation of computational resources this method not performed. Following are the values for temperature at outlet using 'outflow' B.C type.
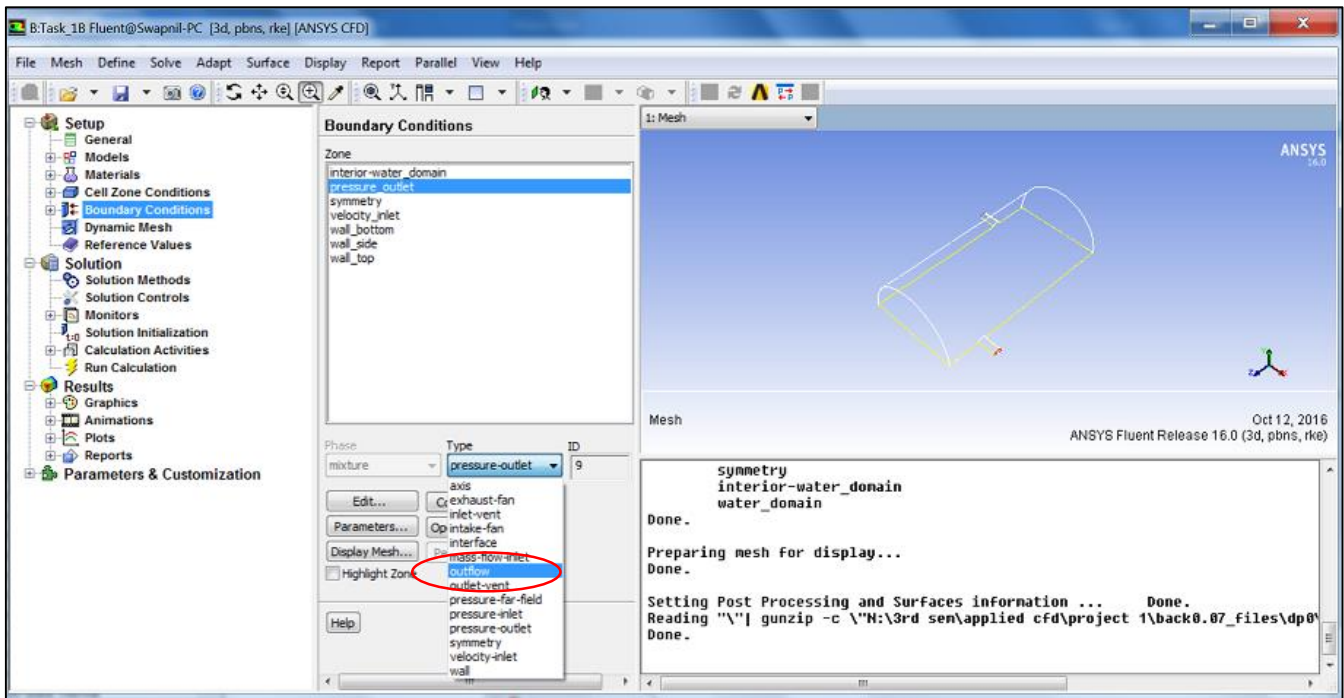
Figure 10 : Window for setting boundary condition type as outflow

After the outflow B.C. the temperature at T_out are as follows for velocity as 0.01 and 0.02.
T_out (0.01) = 326.14 K; T_out (0.02) =320.44 K;
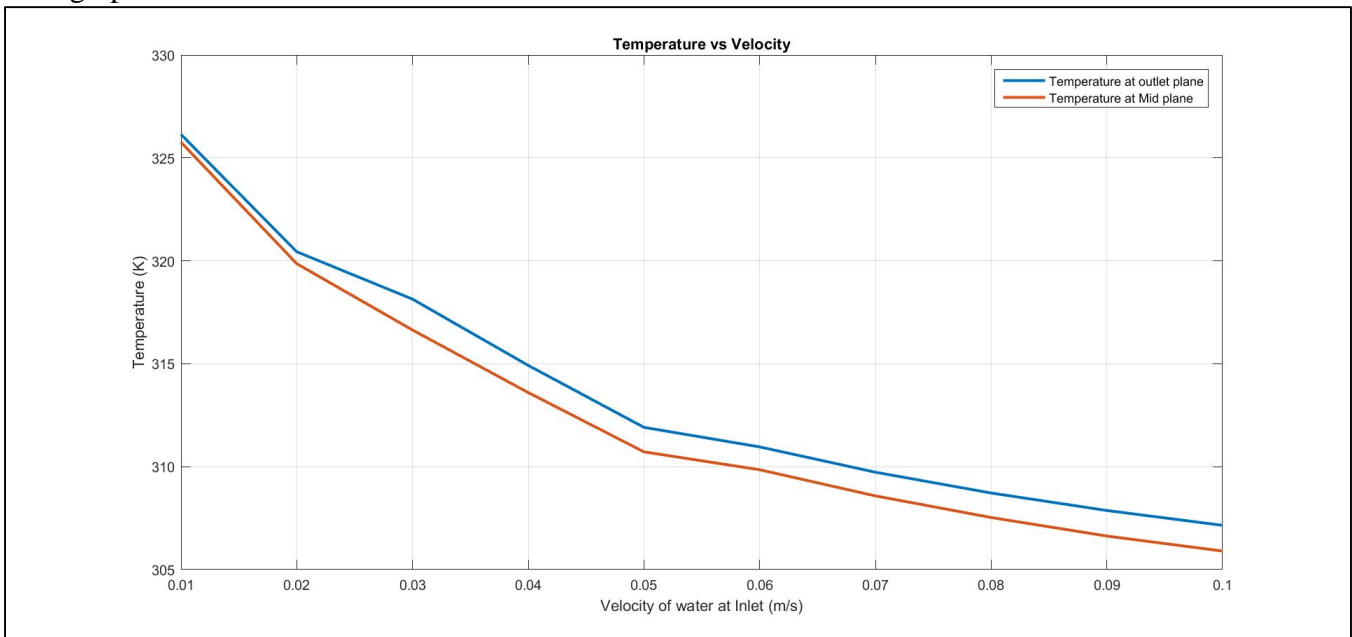
Then graph with these values as shown below

Figure 11 : : Line plot of temperature vs inlet velocity at outlet and mid-plane

Summary: As we increase the inlet velocity there is fall in temperature value at outlet and mid plane. As the velocity increases, water particles remain in tank for less amount of time hence there is less gain in temperature for water particle from conduction. Also as velocity increases the net inlet mass flow also increases which results in reduce the resultant temperature of tank as more cold water is coming inside, which reduces the outlet temperature and mid plane temperature.