MAE 598 Applied CFD

Project 1

Part III: Challenge #2

Gargi Kailkhura (in collaboration with Suhail Jeet Singh)

Using Parametric Design to run 10 simulations

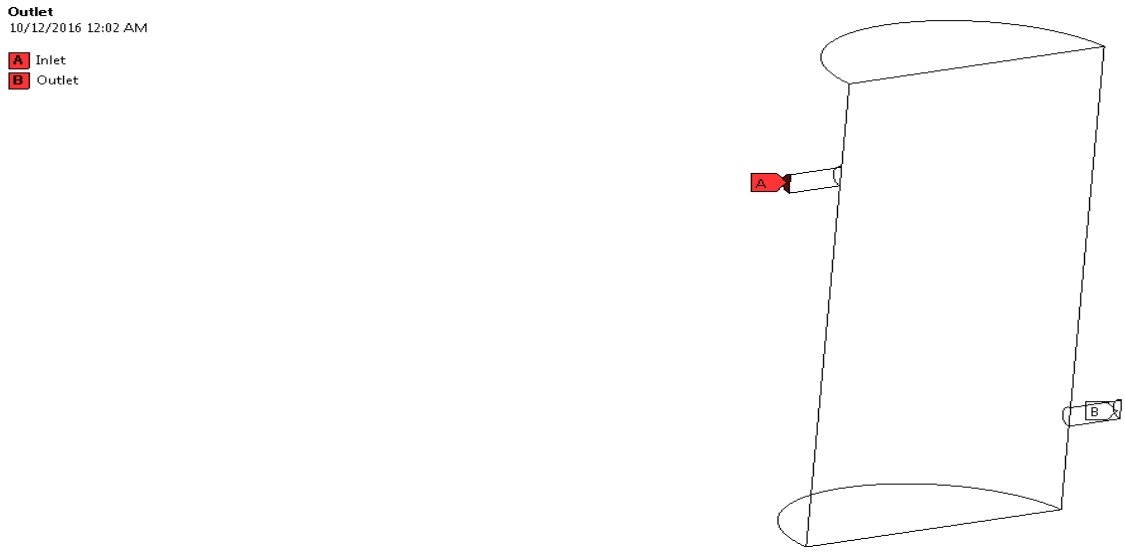


Figure 1: Case-A replica + parametric design

Deliverable-1:

Description for set-up of parametric design

- Step 1: Change in **Boundary Conditions** to set up parameters

Since there are 10 different velocity inputs, so set up the parameters by first **selecting 'new input parameter'** in the velocity type drop-down.

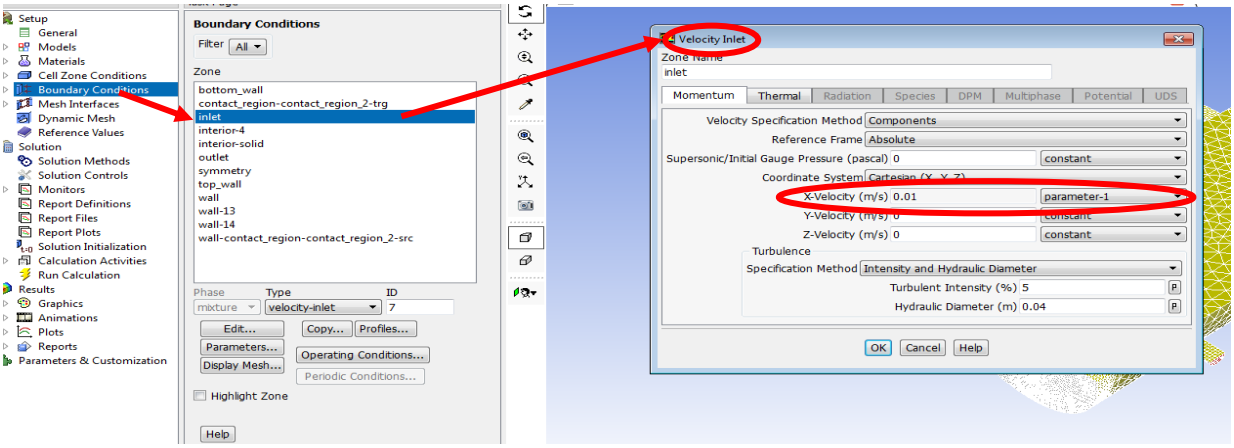


Figure 2: Step 1- Setting up Parameters

- Step 2: Enter **the first Parameter**

One the 'new input parameter' is selected, a dialogue box opens up asking about the properties of the parameter. Enter the first parameter and its value. Click on 'OK' to proceed further.

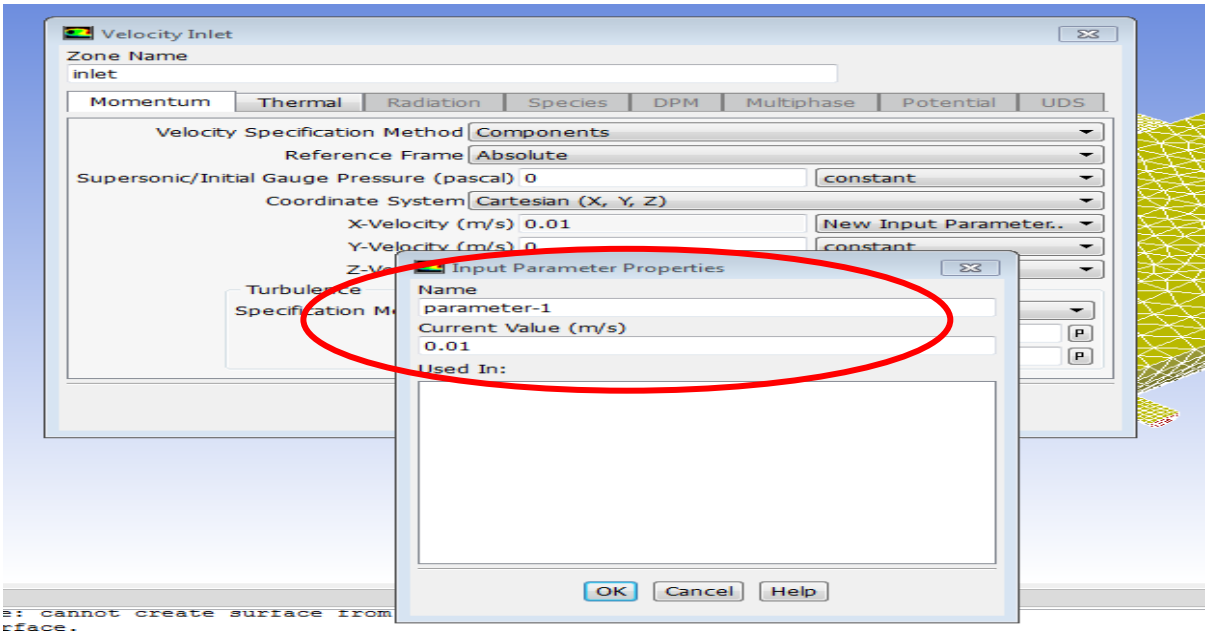


Figure 3: Entering first Parameter values

- Step 3: Create **Output Parameters** in **Parameters and Customization**

In this case, the output parameter is temperature, evaluated using integral formula. So, develop custom field function for the numerator. It is already discussed before in the previous HW1 and Tutorial-2.

$$T_{out} = \iint \frac{v_n * T * dA}{v_n * dA}$$

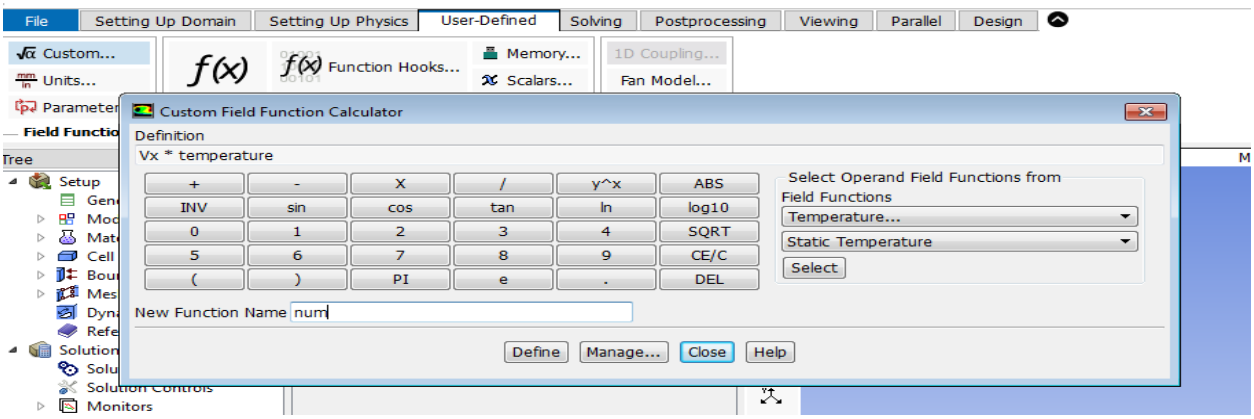
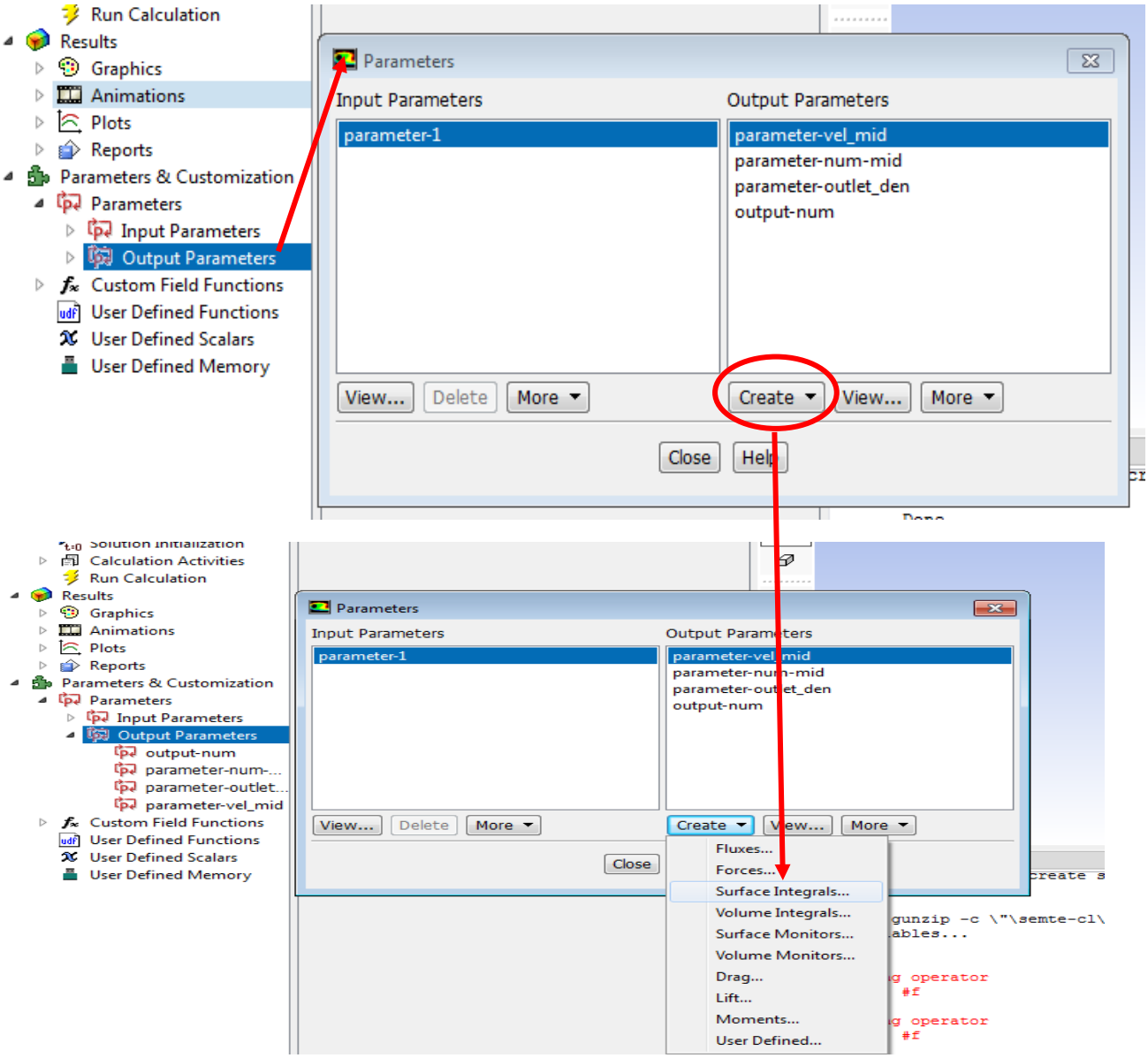


Figure 4: Custom field function for computing temperature

After developing custom field functions, go to '**Parameters and Customization**' in the bottom of the tree. Select output parameters and wait for a new window to open. As seen, parameters are shown classified into input and output. Since, **input parameters** are defined in Step 1 and **Output parameter** (Temperature in this case) is defined just above. Click on '**Create**' to use surface integral option for integrating temperature. Use the surface Integral option as previously used to select field variable (normal velocity and custom field function in this case) and the surface (outlet and z=0.5m plane in this case). Then, click on '**Save Output Parameter**' tab to eventually create the output parameter. One can create as many input or output parameters as wanted.



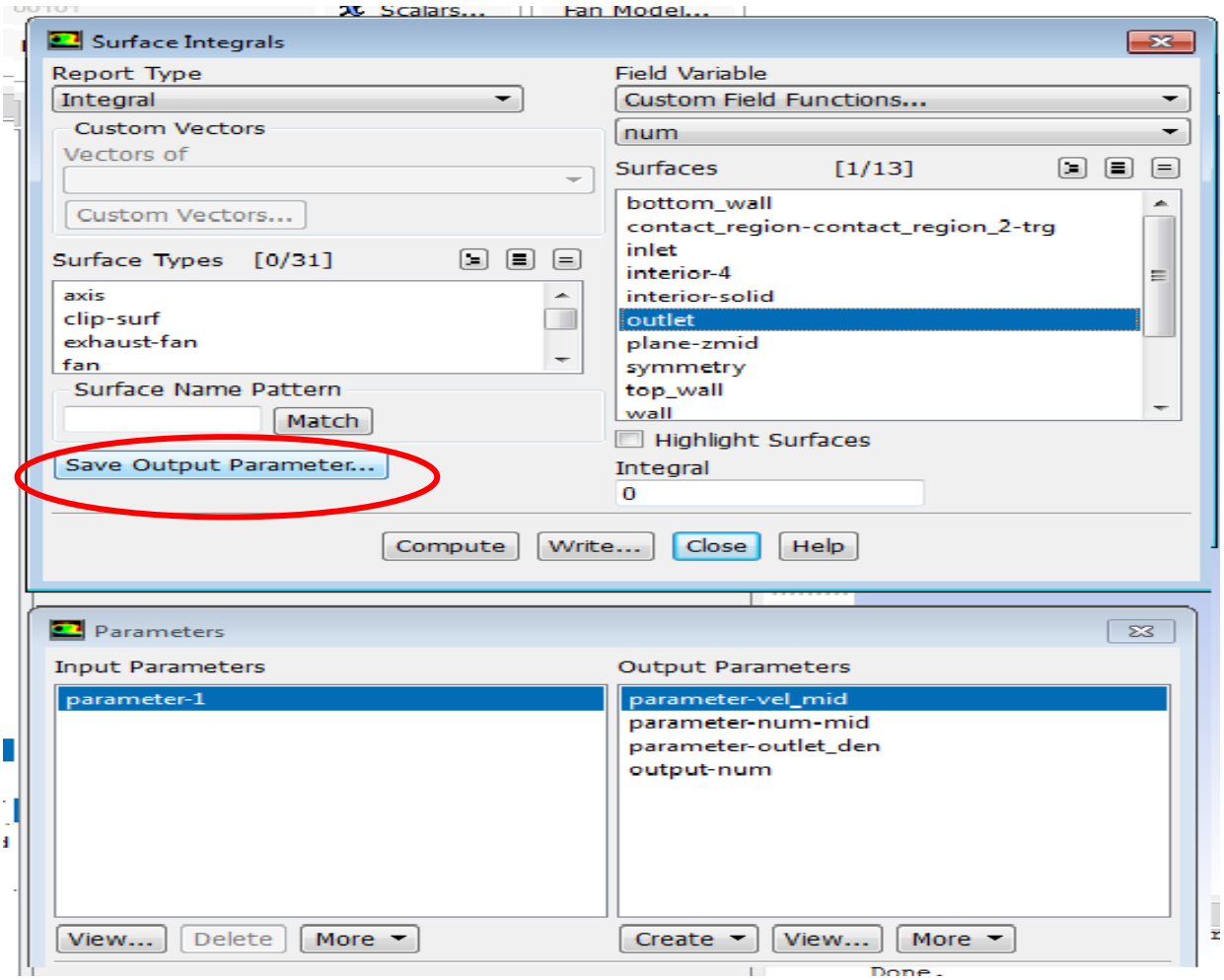
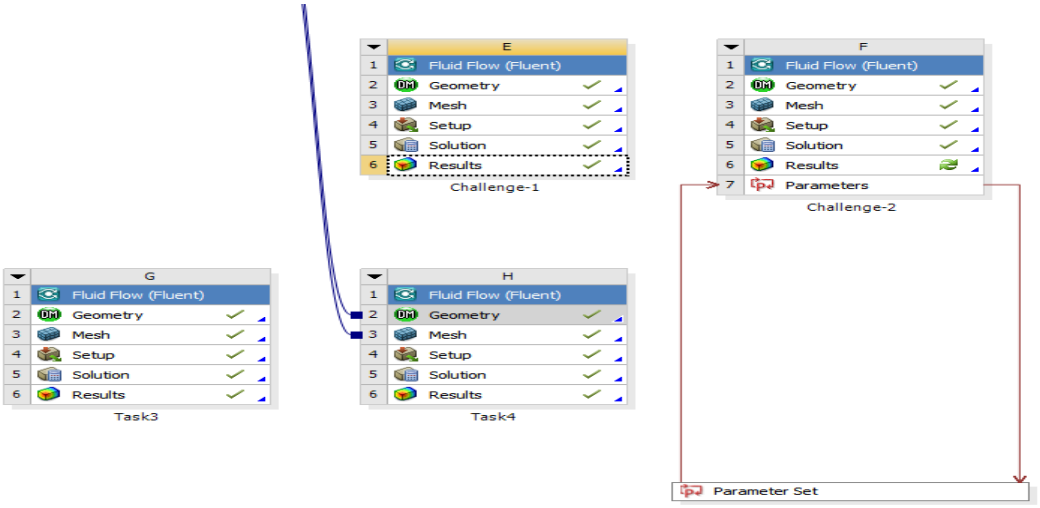


Figure 5: Create Output Parameters

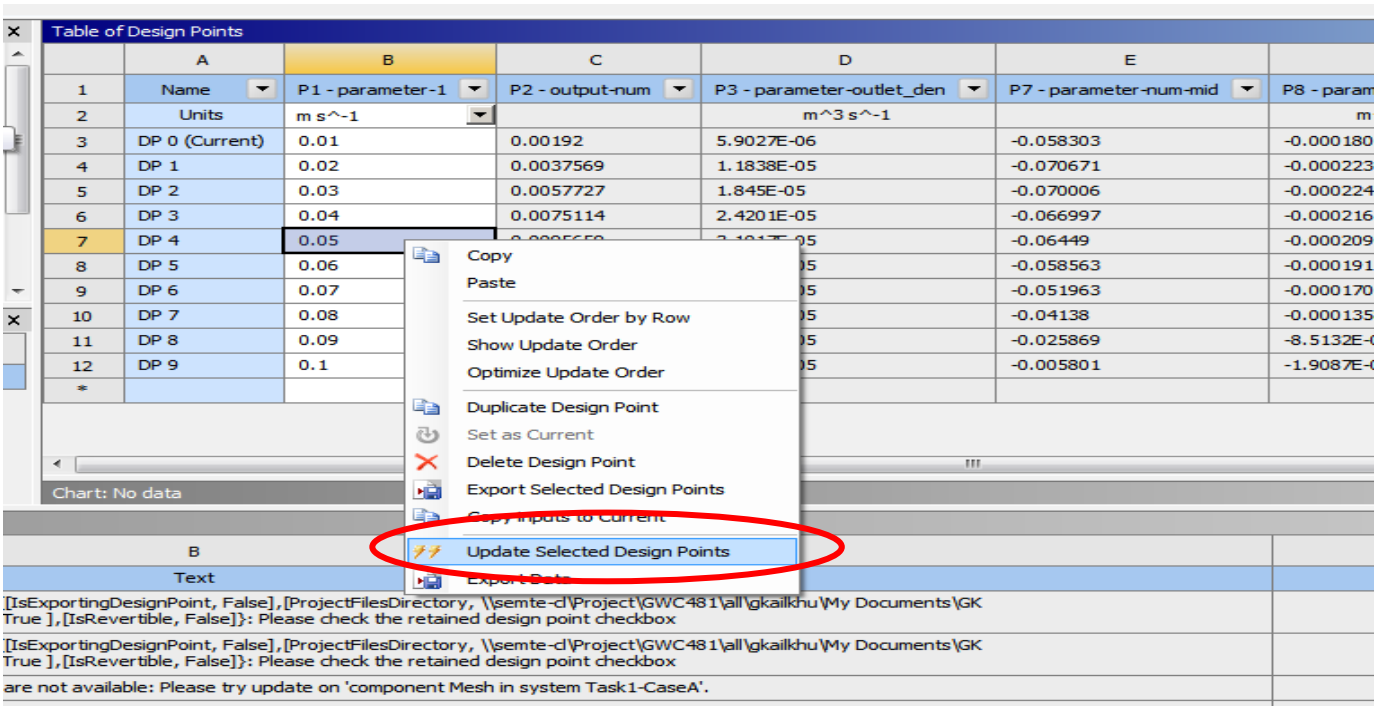
- Step 4: **Initialize** the solution **and run the simulation** normally.
- Step 5: After running the simulation, **go back to the workbench and do not close ANSYS- Fluent**.
- Step 6: In the workbench, **the parameter Set is displayed** across the current Fluid Flow.



- Step 7: Double Click on Parameters to find out that **another tab of Parameters is opened** on the workbench.
- Step 8: **Enter the values of input parameters** (in this case velocity) and right click on the selected input parameter and select the option of '**Update Selected Design Points'**.

Table of Design Points

	A	B	C	D	E	F	G	H	I
1	Name	P1 - parameter-1	P2 - output-num	P3 - parameter-outlet_den	P7 - parameter-num-mid	P8 - parameter-vel_mid	Retain	Retained Data	Note
2	Units	m s^-1		m^3 s^-1		m^3 s^-1			
3	DP 0 (Current)	0.01	0.00192	5.9027E-06	-0.058303	-0.00018053	☑	✓	
4	DP 1	0.02	0.0037569	1.1838E-05	-0.070671	-0.00022342	☐		
5	DP 2	0.03	0.0057727	1.845E-05	-0.070006	-0.00022417	☐		
6	DP 3	0.04	0.0075114	2.4201E-05	-0.066997	-0.00021635	☐		
7	DP 4	0.05	0.0095659	3.1017E-05	-0.06449	-0.00020948	☐		
8	DP 5	0.06	0.011295	3.6772E-05	-0.058563	-0.0001911	☐		
9	DP 6	0.07	0.013069	4.2684E-05	-0.051963	-0.00017016	☐		
10	DP 7	0.08	0.015092	4.9436E-05	-0.04138	-0.00013589	☐		
11	DP 8	0.09	0.016943	5.5615E-05	-0.025869	-8.5132E-05	☐		
12	DP 9	0.1	0.018792	6.1796E-05	-0.005801	-1.9087E-05	☐		
*							☐		



- Step 9: After some time, **output parameters are computed** and are used for post-processing other data. In this case, these parameters are **exported into MATLAB for computing temperature** by dividing numerator by normal component of velocity at both outlet and mid-plane. Line plots are then attained for this processed data.

Deliverable 2:-

Line Plots for Temperature corresponding to different velocity inputs at both outlet and mid-plane are obtained with the help of MATLAB (Step 9).

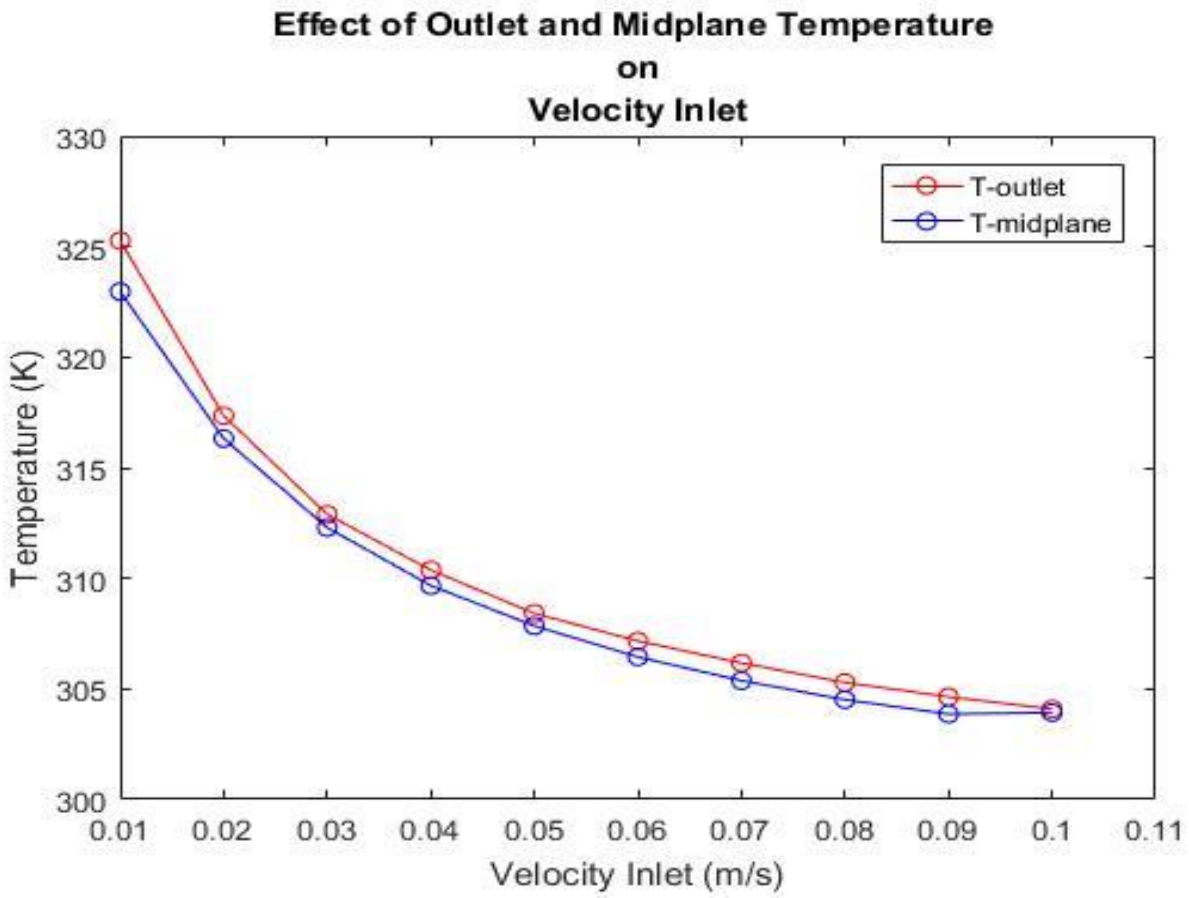


Figure 6: Line Plot of Outlet and Mid-plane Temperature vs Velocity

The outlet temperature is a bit higher than temperature across middle of the tank. Also, temperature values decrease with increase in velocity as smaller the velocity, longer the interaction time of water packets with heat and higher the water temperature.