Using Parametric Design to run 10 simulations

**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.
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.

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.

\[ T_{out} = \iiint \frac{v_n \cdot T \cdot dA}{v_n \cdot dA} \]

Figure 3: Entering first Parameter values

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.
- 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.

**Figure 5: Create Output Parameters**
- 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'.
• 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).

![Line Plot of Outlet and Mid-plane Temperature vs Velocity](image)

*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.