Task 3

In this exercise, we will check the energy balance and mass balance of the system. All the calculations in this task will consider the simulation from Task 1. The flow rate of heat through a given surface, A, is defined by

$$H = \iint v_n \rho \, C_p \, T \, dA \tag{1}$$

where ρ and C_p are density and specific heat of water, v_n is the velocity normal to the surface, and T is temperature. (Thus, H has the unit of power, in $\frac{J}{s}$.) The flow rate of mass through a given surface, A, is defined by
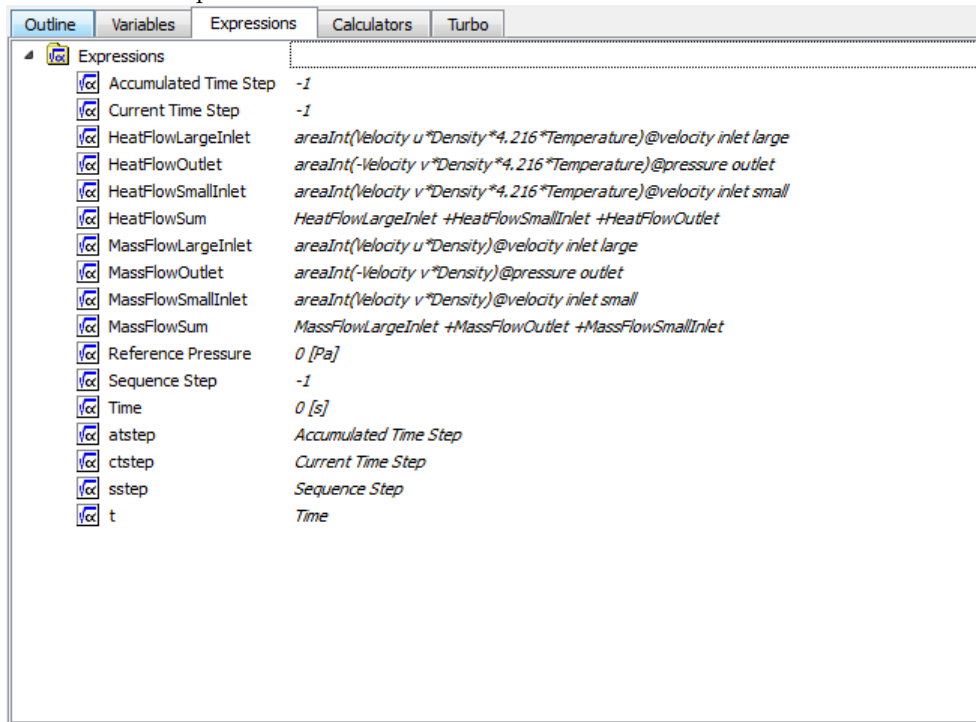
$$M = \iint v_n \rho \, dA \tag{2}$$

(Note that H is the area integral of heat flux while M is the area integral of mass flux.) Since there is no internal heat source or sink in this system, in the steady state the H for the outlet should equal the sum of the H's for the two inlets. Likewise, since the system has no internal mass source/sink, the M for the outlet should equal the sum of the M's for the two inlets. Calculate the three values of H and three values of M to show that the requirements of energy and mass balance are satisfied.

(a) Steps Used to Calculate H, M

In order to make sure that the Heat flow (eq. (1) and and Mass flow (eq. (2)) are satisfied in this simulation we must calculate the above integrals at each inlet and outlet (since there is no internal heat sink, etc.). The way that I did this for this problem was by creating an expression in CFD post (the "results" tab from the general ANSYS workspace). In order to create an expression I used the following steps:

1. Open "results" from ANSYS workspace

2. Click on the "expressions" tab

| Outline	Variables	Expressions	Calculators	Turbo	
▲ Expressions					
	Accumulated Time Step	-1			
	Current Time Step	-1			
	HeatFlowLargeInlet	areaInt(Velocity u*Density*4.216*Temperature)@velocity inlet large			
	HeatFlowOutlet	areaInt(-Velocity v*Density*4.216*Temperature)@pressure outlet			
	HeatFlowSmallInlet	areaInt(Velocity v*Density*4.216*Temperature)@velocity inlet small			
	HeatFlowSum	HeatFlowLargeInlet +HeatFlowSmallInlet +HeatFlowOutlet			
	MassFlowLargeInlet	areaInt(Velocity u*Density)@velocity inlet large			
	MassFlowOutlet	areaInt(-Velocity v*Density)@pressure outlet			
	MassFlowSmallInlet	areaInt(Velocity v*Density)@velocity inlet small			
	MassFlowSum	MassFlowLargeInlet +MassFlowOutlet +MassFlowSmallInlet			
	Reference Pressure	0 [Pa]			
	Sequence Step	-1			
	Time	0 [s]			
	atstep	Accumulated Time Step			
	ctstep	Current Time Step			
	sstep	Sequence Step			
	t	Time			

3. Right click → new

4. Name the new expression (ex. "HeatFlowLargeInlet")

5. You can edit the new expression in the lower window under the "definition" tab
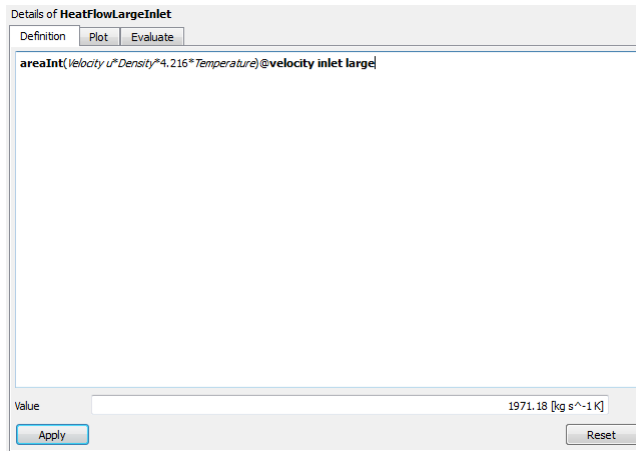
6. To create an integral

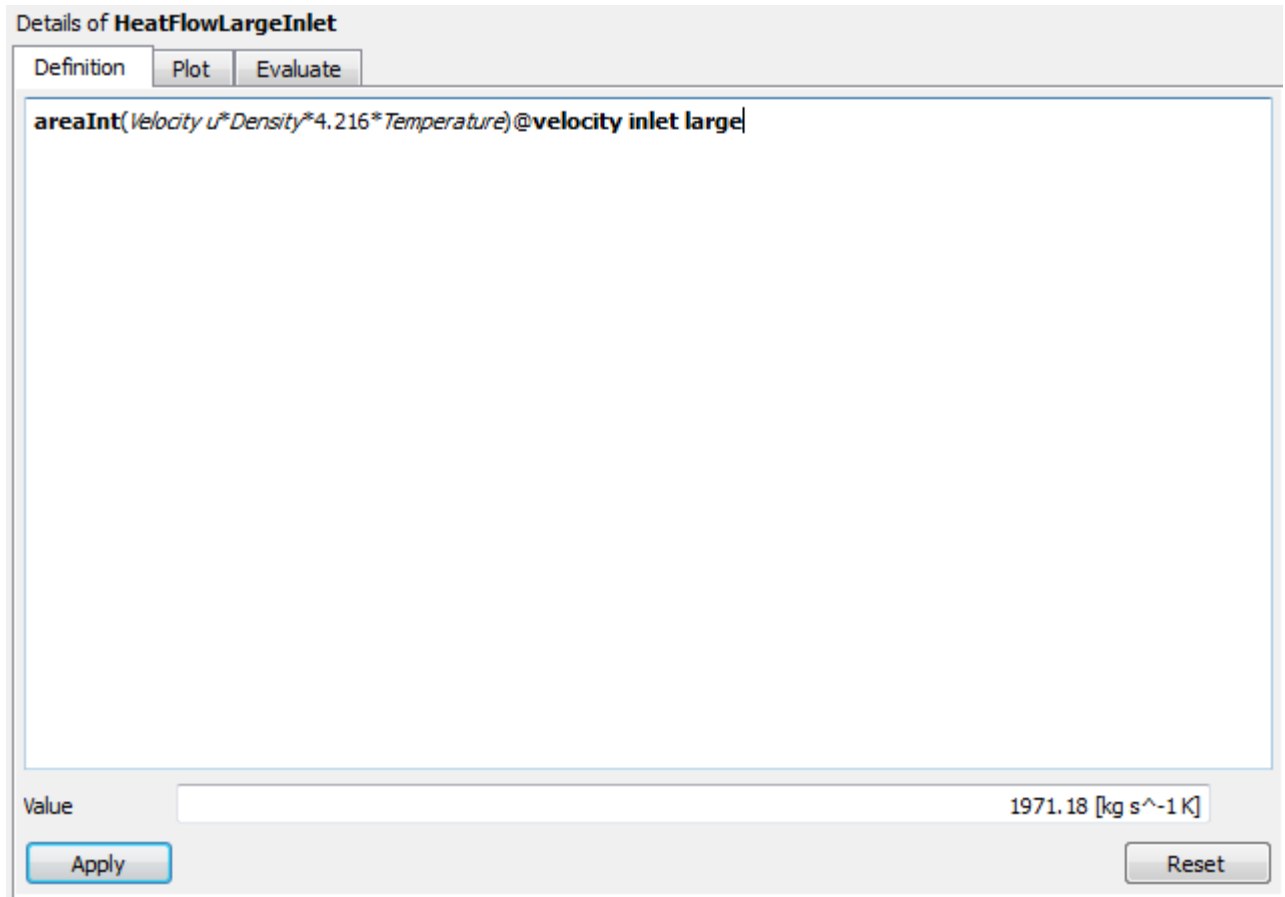   right click → functions → CFD-post → "areaint"
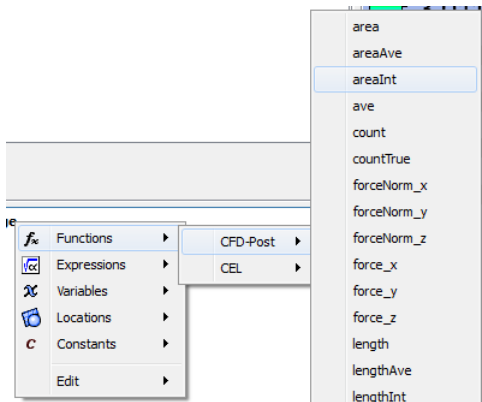
7. This will create a format for you to input the expression to be integrated at the location (surface) that is preferred. The format after you insert the areaint will look like this: "areaint()@." The integrand goes in the parentheses. We want to put the proper location of the integral after the @ sign.

   right click (after the @ sign) → locations → "location" (ex. "Velocity Inlet large")
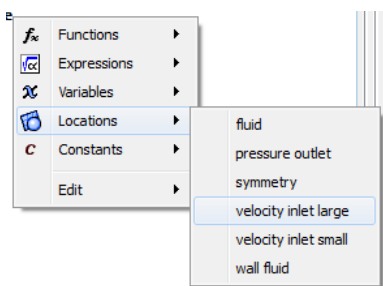
8. Enter the expression from either eq. (1) or (2) depending on which one you are calculating, and make sure it is entered properly. To enter variables like "velocity" and "density," etc:

right click → variables → "variable name" (ex "velocity u")

Details of **HeatFlowLargeInlet**

| Definition | Plot | Evaluate |

areaInt(*Velocity u*Density*4.216*Temperature*)@**velocity inlet large**

Value   1971.18 [kg s^-1 K]

Apply   Reset

9. Click apply (The "value" field should populate with the answer.)

10. Repeat the steps for all three inlets and outlets and for both integral equations. You should have 6 equations total. 8 if you want to enter expressions for the sum of all 3 inlets and outlets for both M and H. But this is not necessary. You can sum them later with excel or a calculator.

(b) Results

Below are the results I found using CFD-post results.

HeatFlowLargeInlet	1971.18	KJ/s		Cp	4216 J/kg-K
HeatfFlowSmallInlet	789.625	KJ/s			
HeatFlowOutlet	-2760.81	KJ/s			
MassFlowLargeInlet	1.59491	kg/s			
MassFlowSmallInlet	0.598092	kg/s			
MassFlowOutlet	-2.193	kg/s			
HeatFlowSum	-0.000976563	KJ/s			
MassFlowSum	0	kg/s			

Using the following expressions

$\sqrt{x}$	HeatFlowLargeInlet	*areaInt(Velocity u*Density*4.179*Temperature)@velocity inlet large*
$\sqrt{x}$	HeatFlowOutlet	*areaInt(-Velocity v*Density*4.179*Temperature)@pressure outlet*
$\sqrt{x}$	HeatFlowSmallInlet	*areaInt(Velocity v*Density*4.179*Temperature)@velocity inlet small*
$\sqrt{x}$	HeatFlowSum	*HeatFlowLargeInlet +HeatFlowSmallInlet +HeatFlowOutlet*
$\sqrt{x}$	MassFlowLargeInlet	*areaInt(Velocity u*Density)@velocity inlet large*
$\sqrt{x}$	MassFlowOutlet	*areaInt(-Velocity v*Density)@pressure outlet*
$\sqrt{x}$	MassFlowSmallInlet	*areaInt(Velocity v*Density)@velocity inlet small*
$\sqrt{x}$	MassFlowSum	*MassFlowLargeInlet +MassFlowOutlet +MassFlowSmallInlet*

note: I took this screen-shot before I changed C_p to the proper value of 4.216 $\left[\frac{KJ}{kg-K}\right]$

The above calculations are consistent with what we would expect. Obviously, both Heat flux and Mass flux of the system should be exactly zero, and this is the case that we see for mass flux. This is expected also, because ANSYS uses the conservation of mass equations in it's process of running the simulation. So usually, ANSYS will have results that always EXACTLY satisfy the conservation of mass. However, Heat flux is just above zero because of the assumption that we made about constant specific heat $\left(C_p = 4216 \left[\frac{J}{kg-K}\right]\right)$. We know that temperature is changing throughout time and location of the pipe, so specific heat should be a function of temperature, rather than a constant. It is possible to create a function that calculates the specific heat as a function of temperature by writing a UDF (user-defined-function) in C++ and then uploading that file to a UDF in the CFD-pre portion of the simulation/analysis, but this proved to complicated, and a constant specific heat was used instead. This is why the calculation is non-zero. However it is close enough to zero (only 0.98 $\left[\frac{J}{kg-K}\right]$) that it is consistent with our expectations.