

PROCESS SIMULATION LAB

Day 2 Group 2

Student's Name	Roll no
Prince Yadav	002010301008
Shreya Ghosh	002010301009
Soumodip Paul	002010301010
Shayantan Sahoo	002010301011

Submission Date

16/08/2023

Question 1

Steam at 500 *bar* and 600 °C is allowed to undergo a Joule Thomson through an adiabatic valve to atmospheric pressure. What will be the temperature of steam after the expansion? Use the ASME steam package.

Answer


Flow Basis 100 *kg/hour*

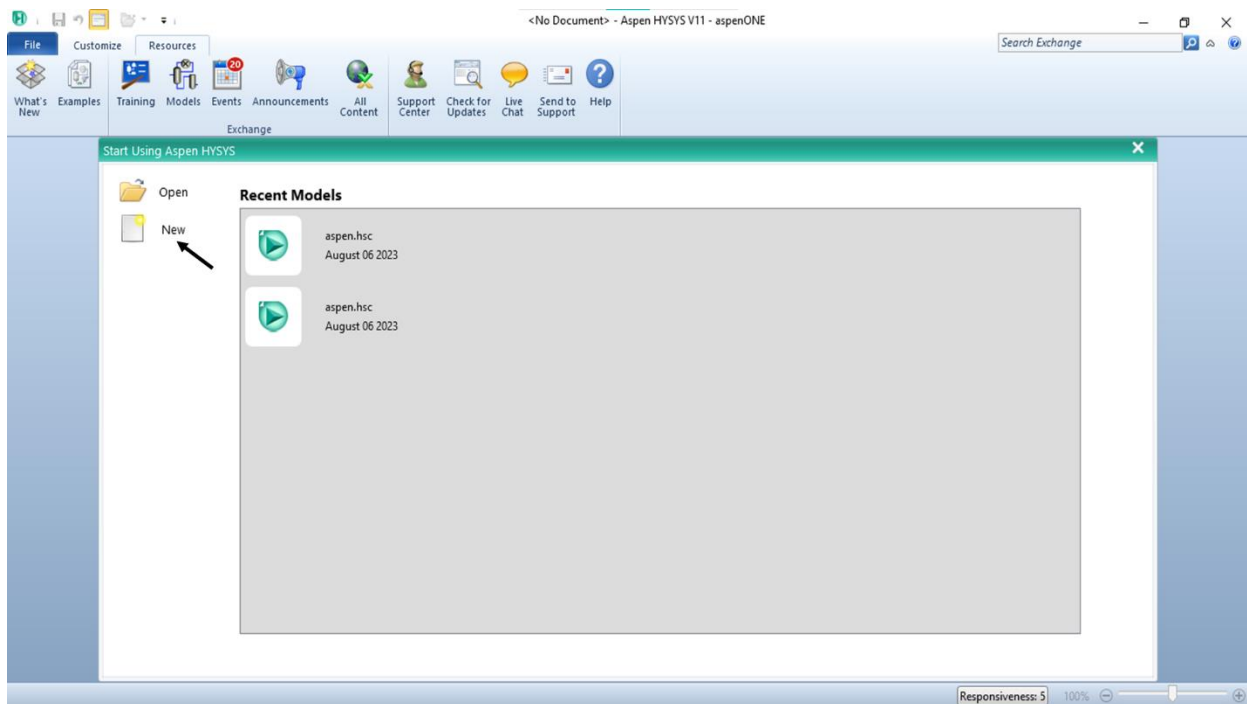
Intet			VLV-100			Outlet		
Temperature	600.0	C				Temperature	385.5	C
Pressure	5.000e+004	kPa				Pressure	101.3	kPa
Molar Flow	55.51	kgmole/h				Molar Flow	55.51	kgmole/h
Mass Enthalpy	-1.268e+004	kJ/kg				Mass Enthalpy	-1.268e+004	kJ/kg

The outlet temperature of the Stream will be 385.5 °C.


Steps to solve the questions are as follows:

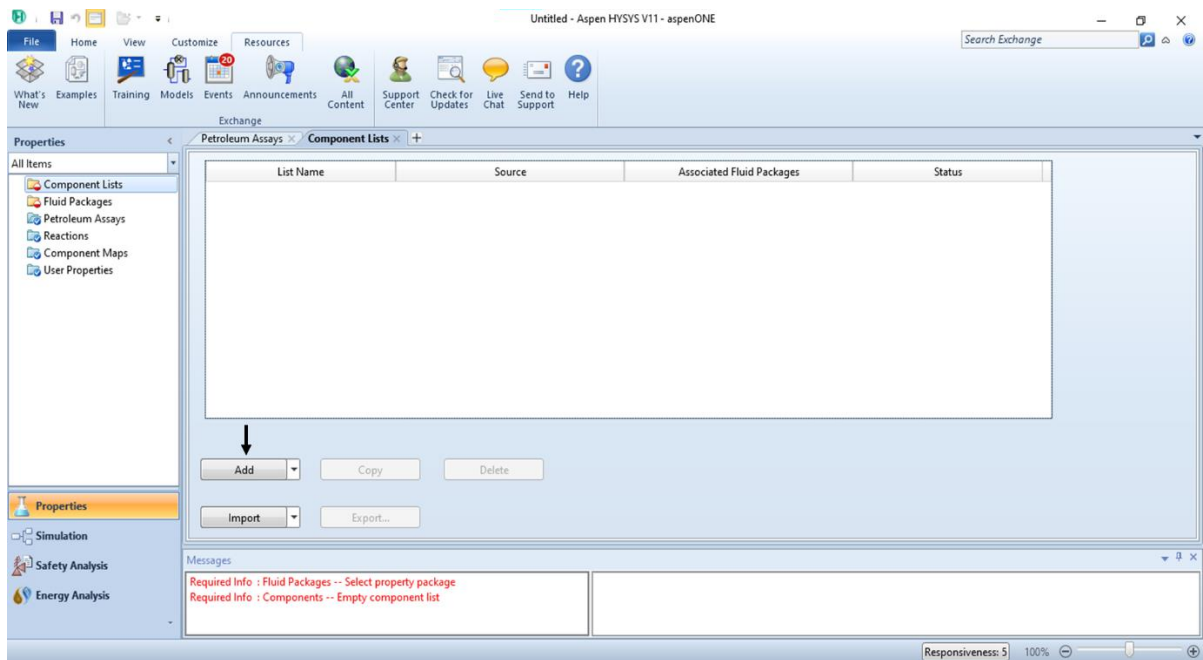
Step 1:

At first, we ASPEN HYSYS software by clicking on the shortcut icon from the desktop. The Initial Layout looks like the following. From the  New menu we clicked on button to create a blank Simulation Workbook page.


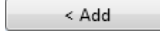



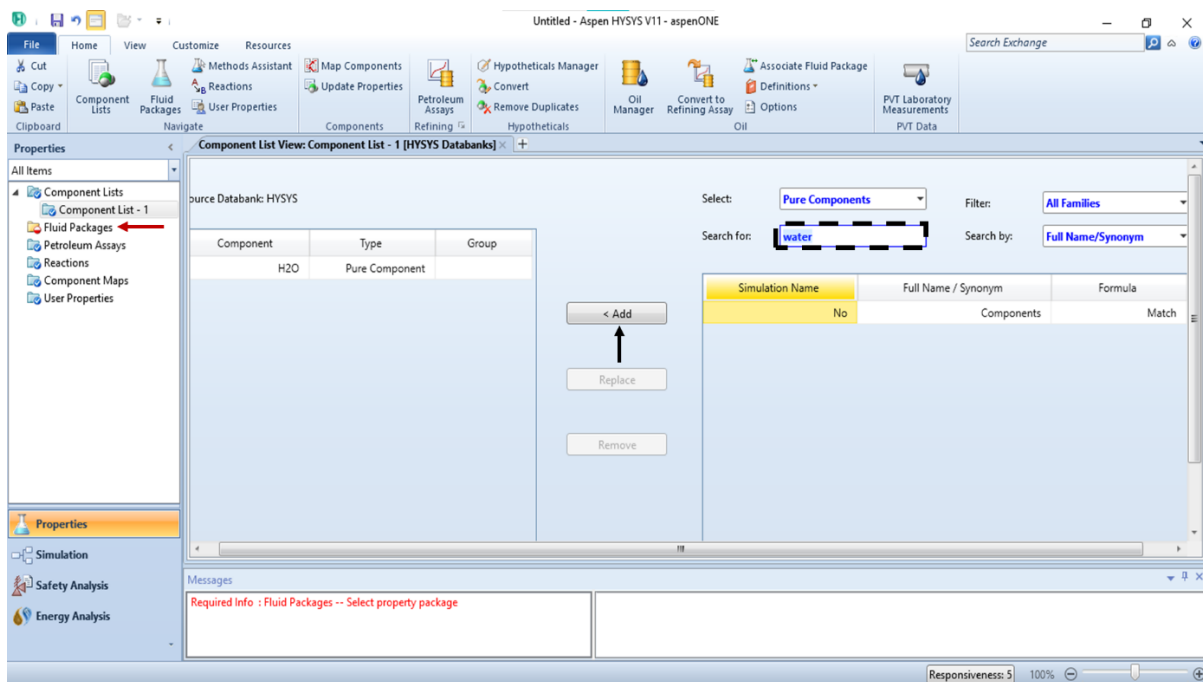
Step 2:

Now in the next page we will click on  icon to select the components required for our simulation. Here we are dealing only with water we will add only water.



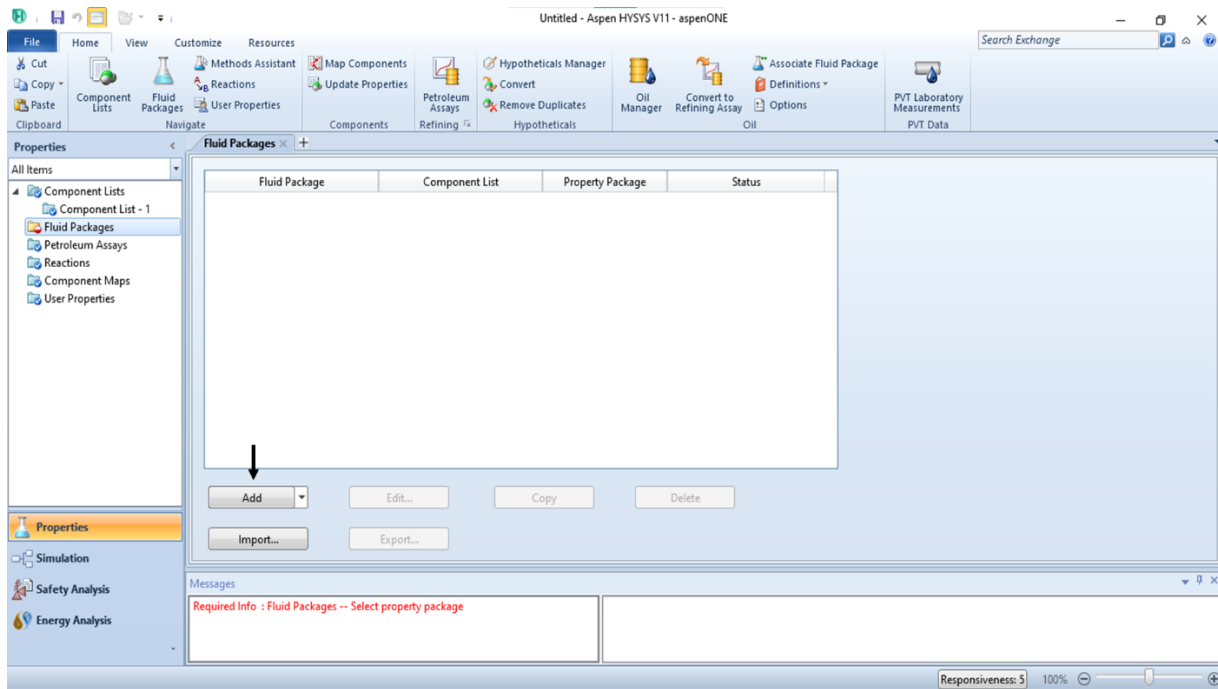
Step 4:

Now in the Component list page we will search water in  box. Then we will select the water and click.  This will add water in the component list. Now we will choose the desired fluid packages by clicking on  Fluid Packages icon.




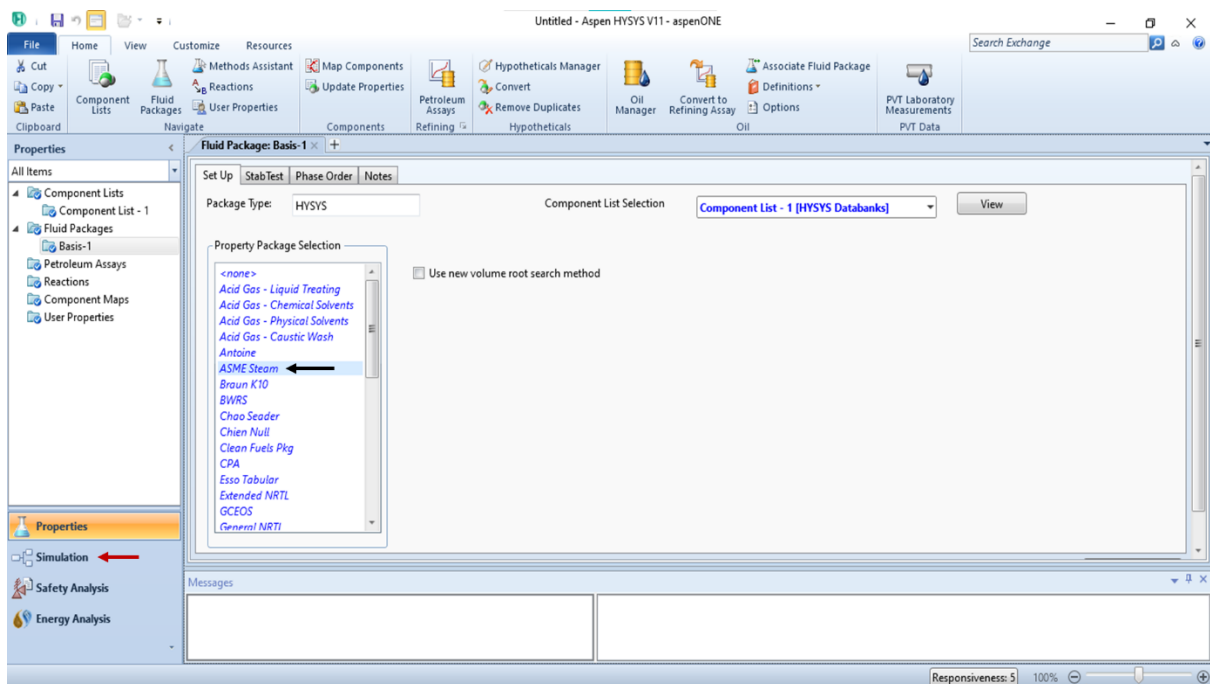
Step 5:

In the Fluid Packages tab, we will click  to add new fluid packages.



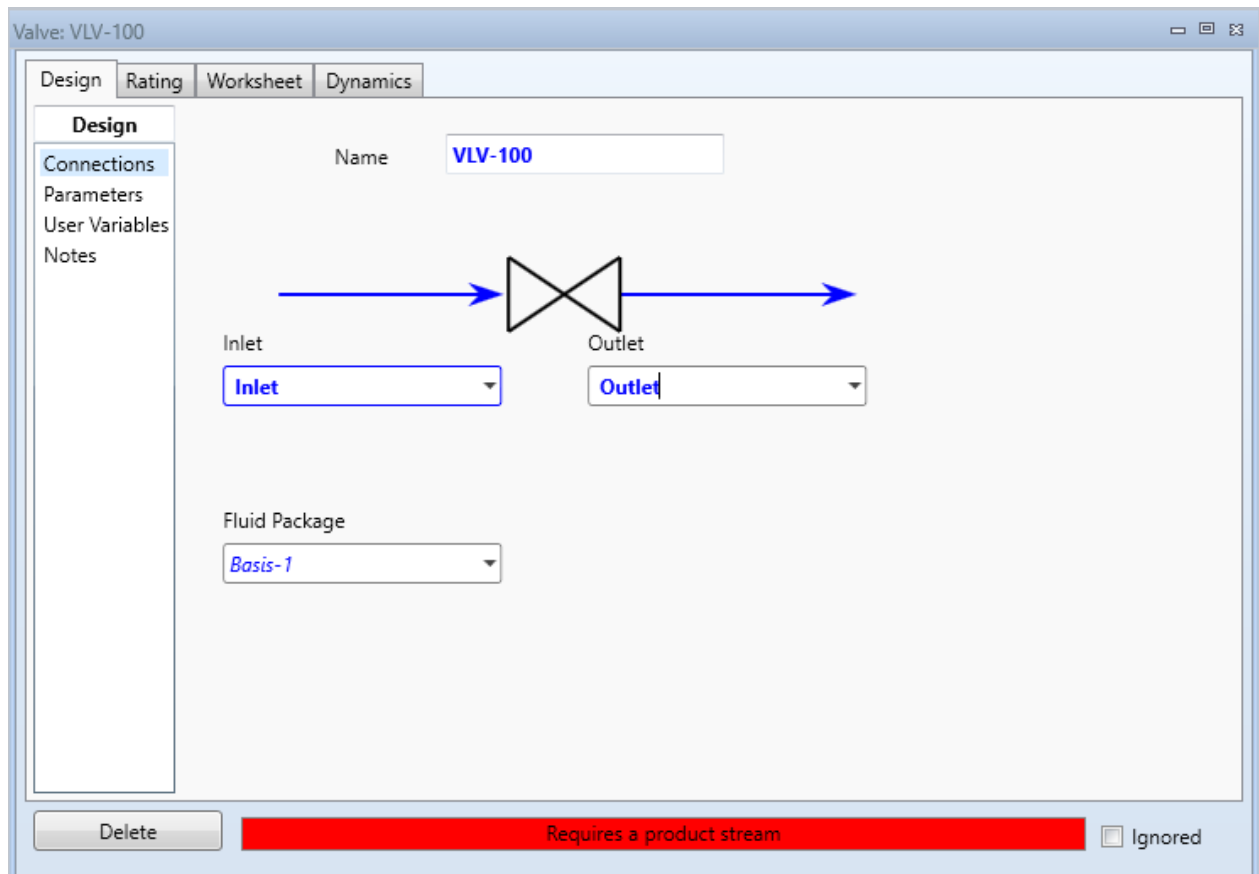
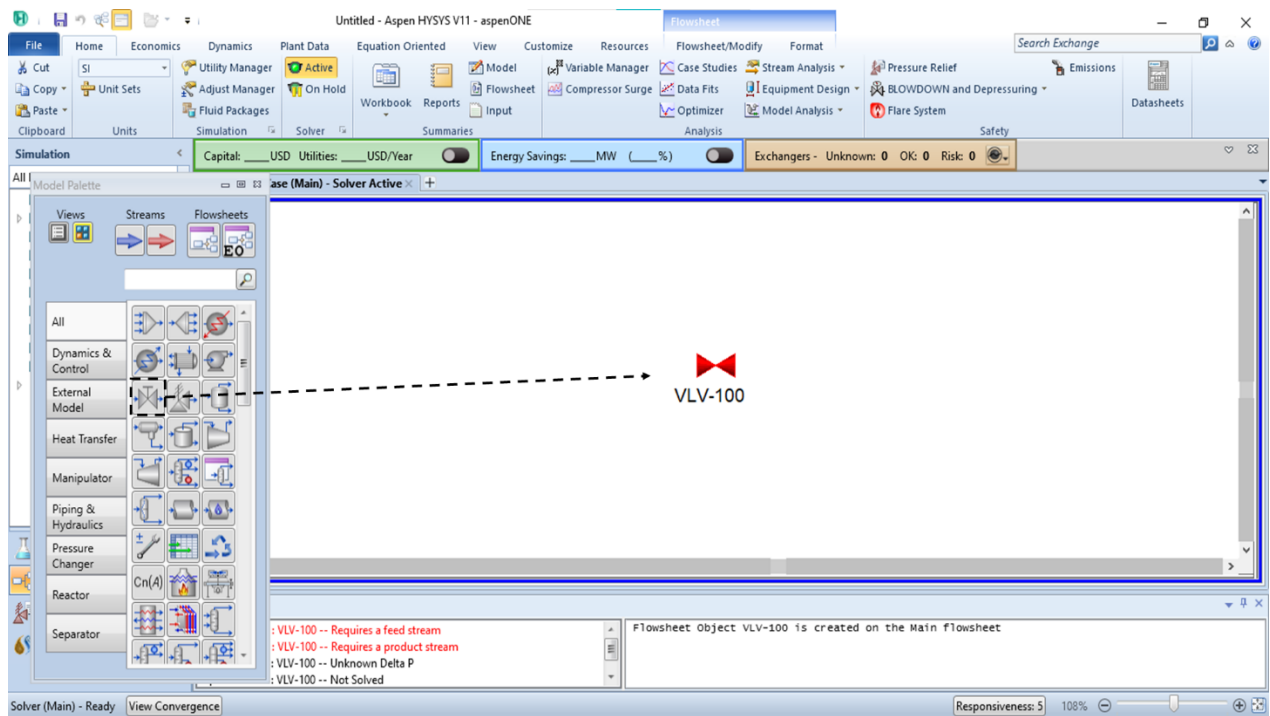
Step 6:

According to our given problem, we are supposed to use ASME steam package for our problem. So, from the dropdown menu, we will select the ASME steam option. Now our components and property databases are ready. We are ready to move to Simulation Tab by clicking  **Simulation** the button.



Step 7:

This is the most important step. First, we will drag the control valve and drop it to the blank space. Then, we will click on  icon. It will open a system dialogue where we will enter the input and output stream names. The colour bar is red as we have not entered any data yet.



Now we will click **Worksheet** on tab which will open another dialogue where we can enter the inlet and outlet stream properties according to the given problem statements.

Valve: VLV-100

Design Rating Worksheet Dynamics

Worksheet

Conditions Properties Composition PF Specs

Name	Inlet	Outlet
Vapour	<empty>	<empty>
Temperature [C]	600.0	<empty>
Pressure [kPa]	5.000e+004	101.3
Molar Flow [kgmole/h]	<empty>	<empty>
Mass Flow [kg/h]	100.0	100.0
Std Ideal Liq Vol Flow [m3/h]	<empty>	<empty>
Molar Enthalpy [kJ/kgmole]	<empty>	<empty>
Molar Entropy [kJ/kgmole-C]	<empty>	<empty>
Heat Flow [kJ/h]	<empty>	<empty>

Delete Not Solved Ignored

Now, the colour bar is yellow as our solution is not converged yet, as we have not yet entered the composition of the water (which is actually 1 as it is pure water). So, we will double click on **Molar Flow [kgmole/h]** tab which will open another window where we can enter the composition, click and close the window.

Input Composition for Stream: Material Stream: Outlet

	Mole Fraction
H2O	1.0000

Composition Basis

☒ Mole Fractions

☐ Mass Fractions

☐ Liq Volume Fractions

☐ Mole Flows

☐ Mass Flows

☐ Liq Volume Flows

Composition Controls

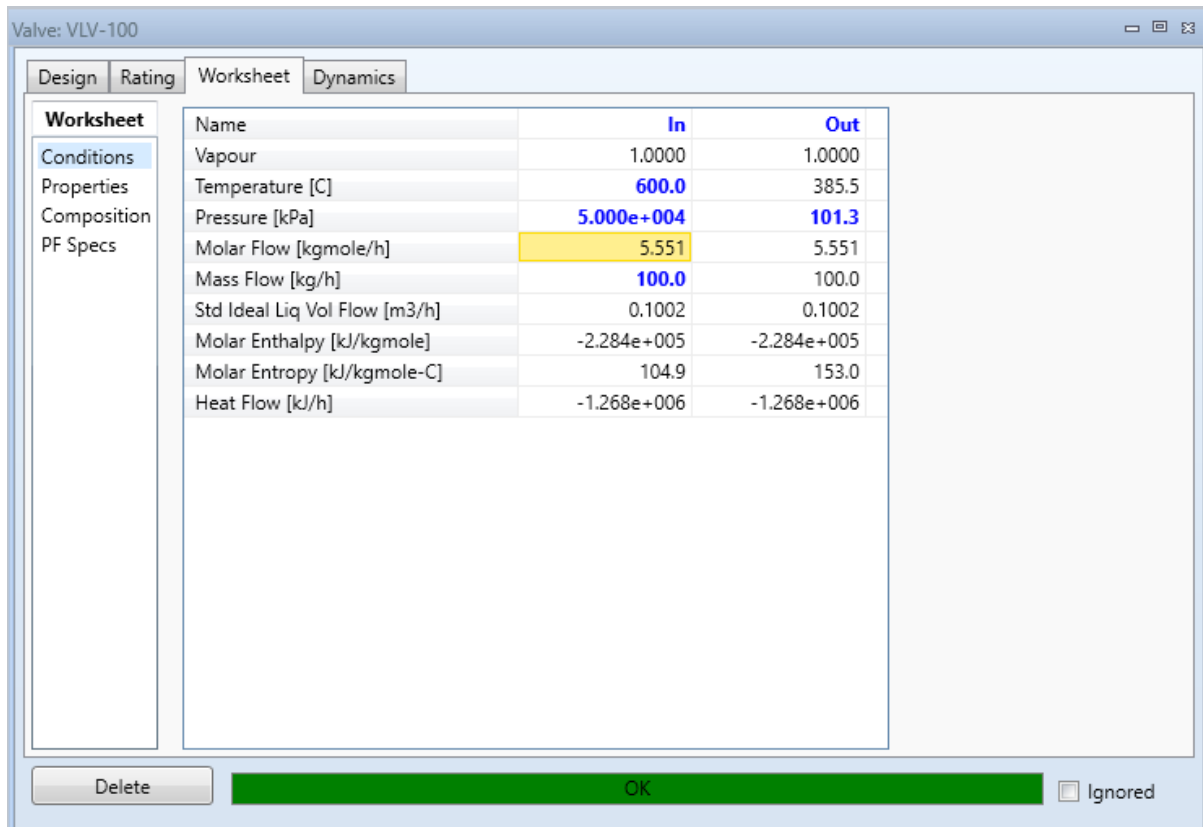
Erase

Equalize Composition

Cancel

Normalize Total 1.0000 OK

Now the colour bar turns green which means our simulation has been converged.



Name	In	Out
Vapour	1.0000	1.0000
Temperature [C]	600.0	385.5
Pressure [kPa]	5.000e+004	101.3
Molar Flow [kgmole/h]	5.551	5.551
Mass Flow [kg/h]	100.0	100.0
Std Ideal Liq Vol Flow [m3/h]	0.1002	0.1002
Molar Enthalpy [kJ/kgmole]	-2.284e+005	-2.284e+005
Molar Entropy [kJ/kgmole-C]	104.9	153.0
Heat Flow [kJ/h]	-1.268e+006	-1.268e+006

After than we can close the window and go back to our simulation page. Now we will right click on the valve and streams and click on the show table option to show necessary outputs. Our output is shown at the very beginning of the report.