PROCESS SIMULATION LAB

Day 5
Group 2

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

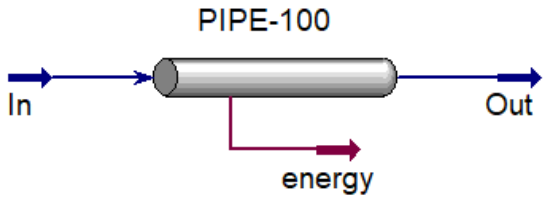
Submission Date

13/09/2023

Question 1

Waters enters a 5000 ft long 6 in diameter schedule 40 commercial steel pipe at 60 °F and 150 $psig$ and discharges to an open tank located 300 ft above the inlet point. Calculate the discharge water flow rate. Assume no heat loss from the pipe.

Answer



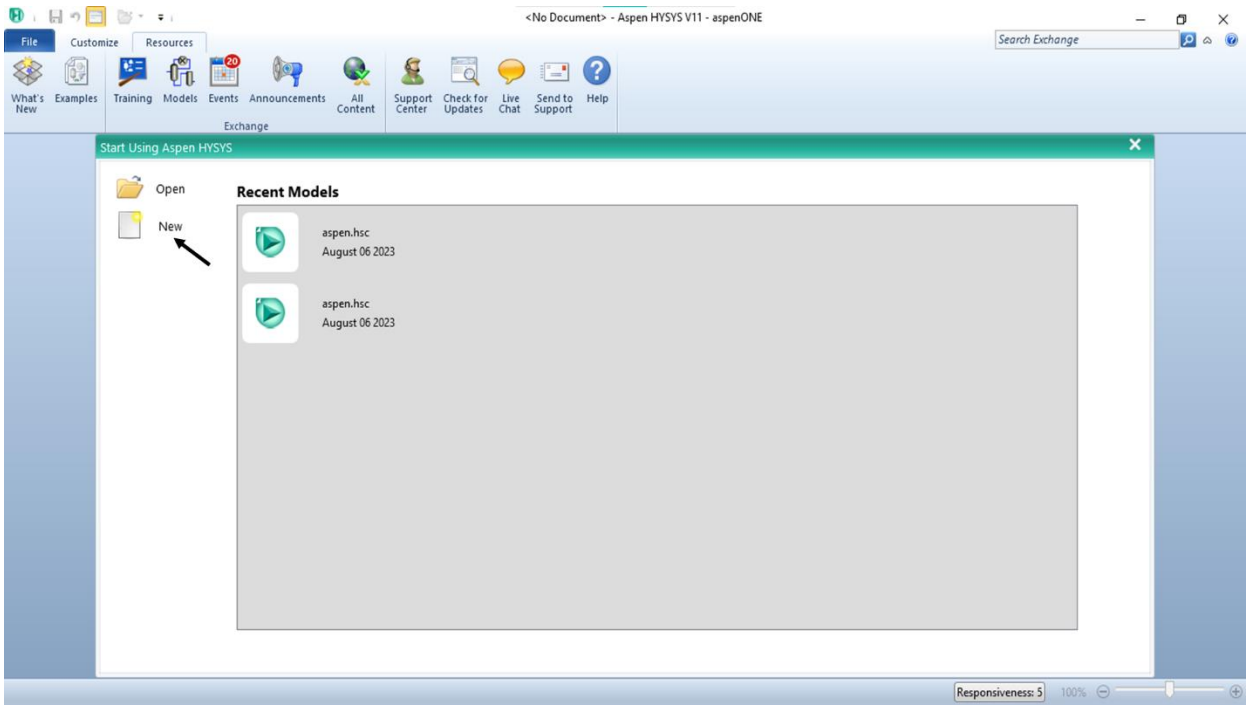
PIPE-100		
Inside Diameter(1)	6.065	in
Outside Diameter(1)	6.625	in
Pipe length(1)	5000	ft
Elevation(1)	300.0	ft
Feed Temperature	60.00	F
Feed Pressure	164.7	psia
Product Pressure	14.70	psia
Overall Pressure Drop	150.0	psi
Volume Flow	367.7	USGPM

The outlet flowrate of the water is 367.7 gpm.

Steps to solve the questions are as follows:

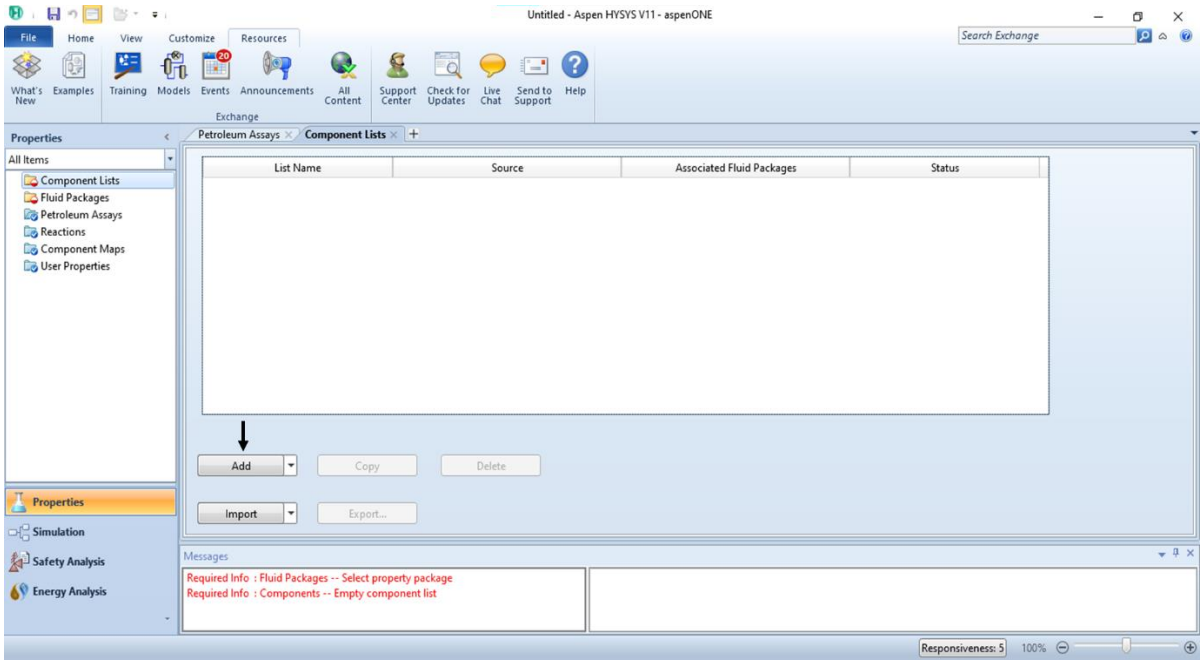
Step 1:

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.



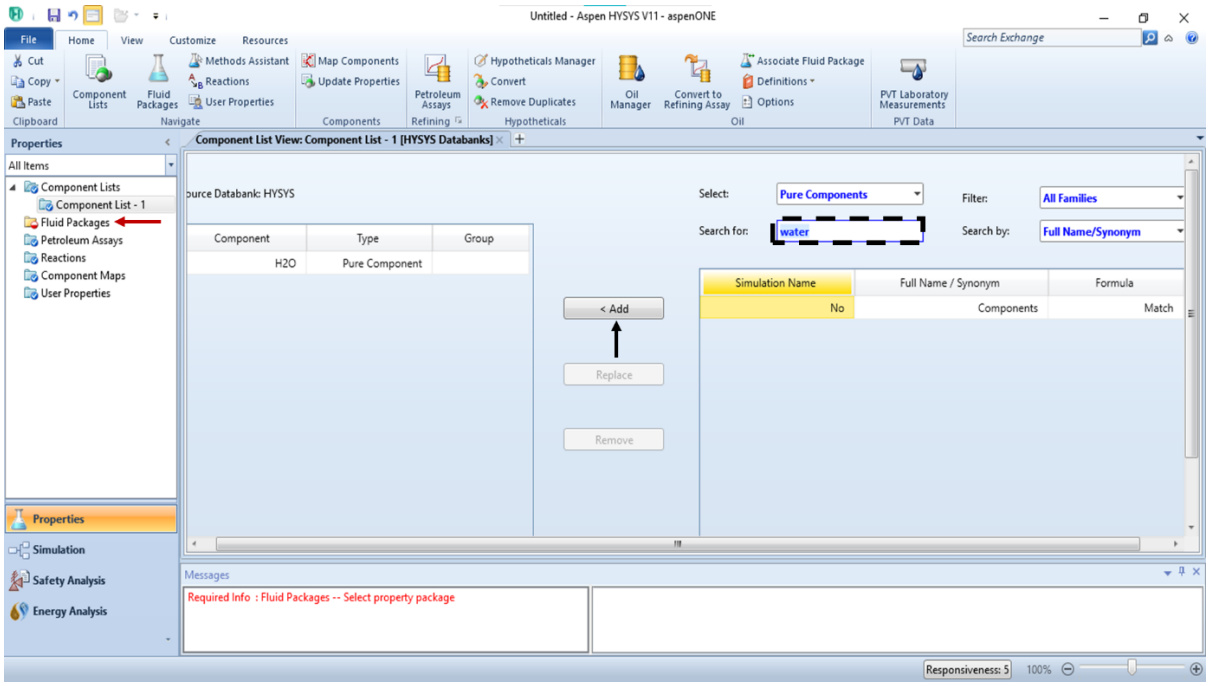
Step 2:

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



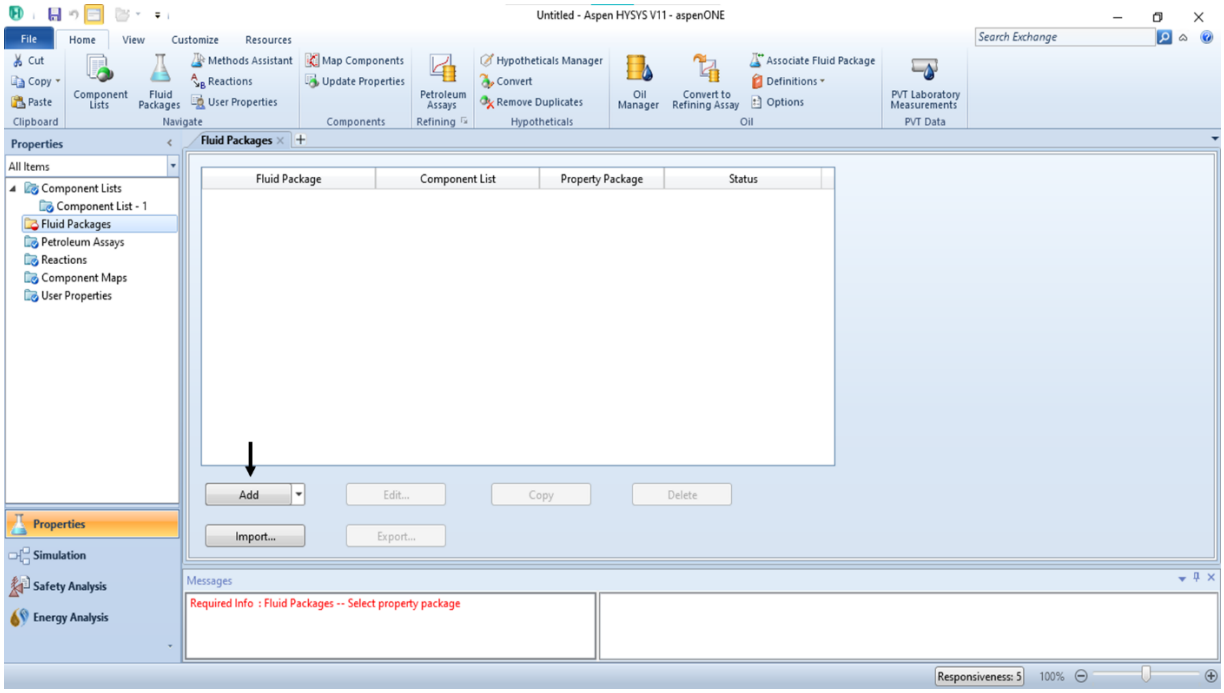
Step 4:

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

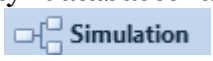


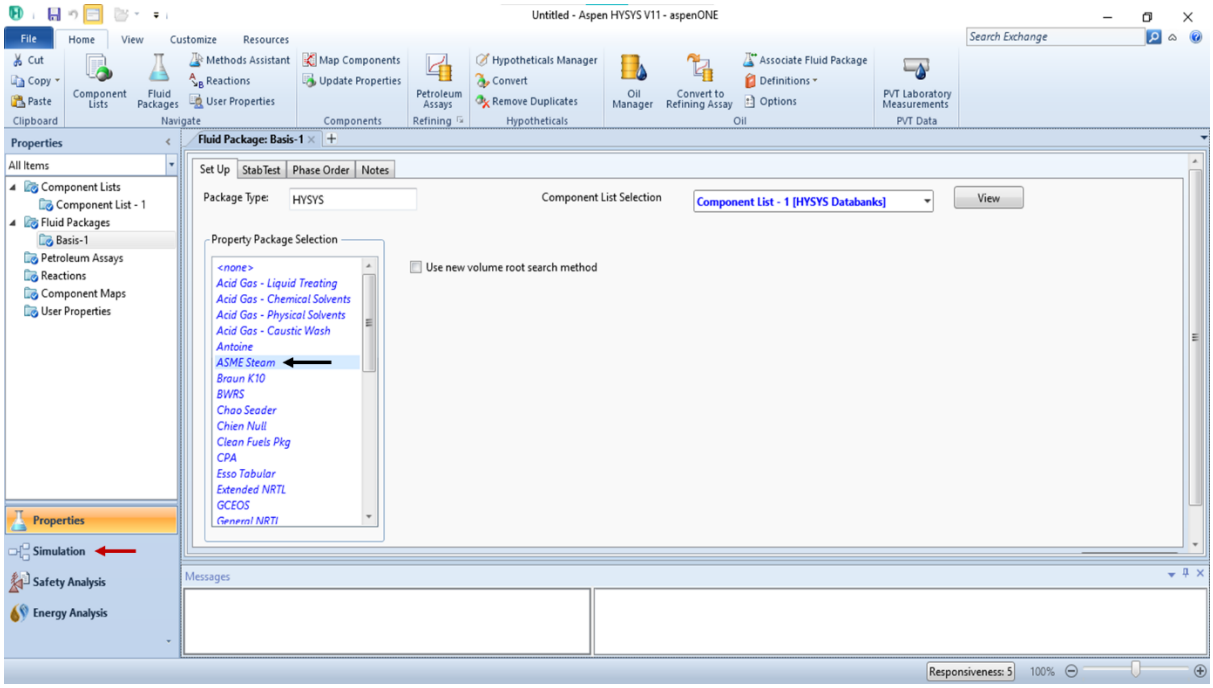
Step 5:

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



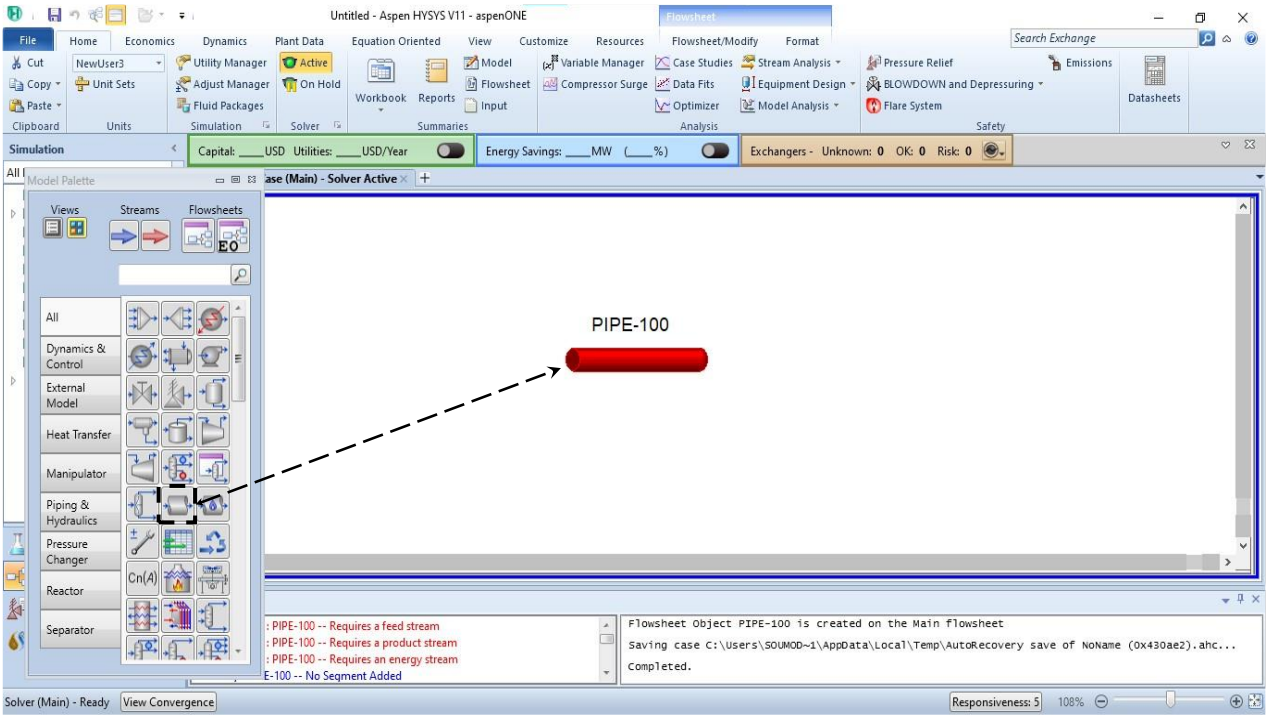
Step 6:

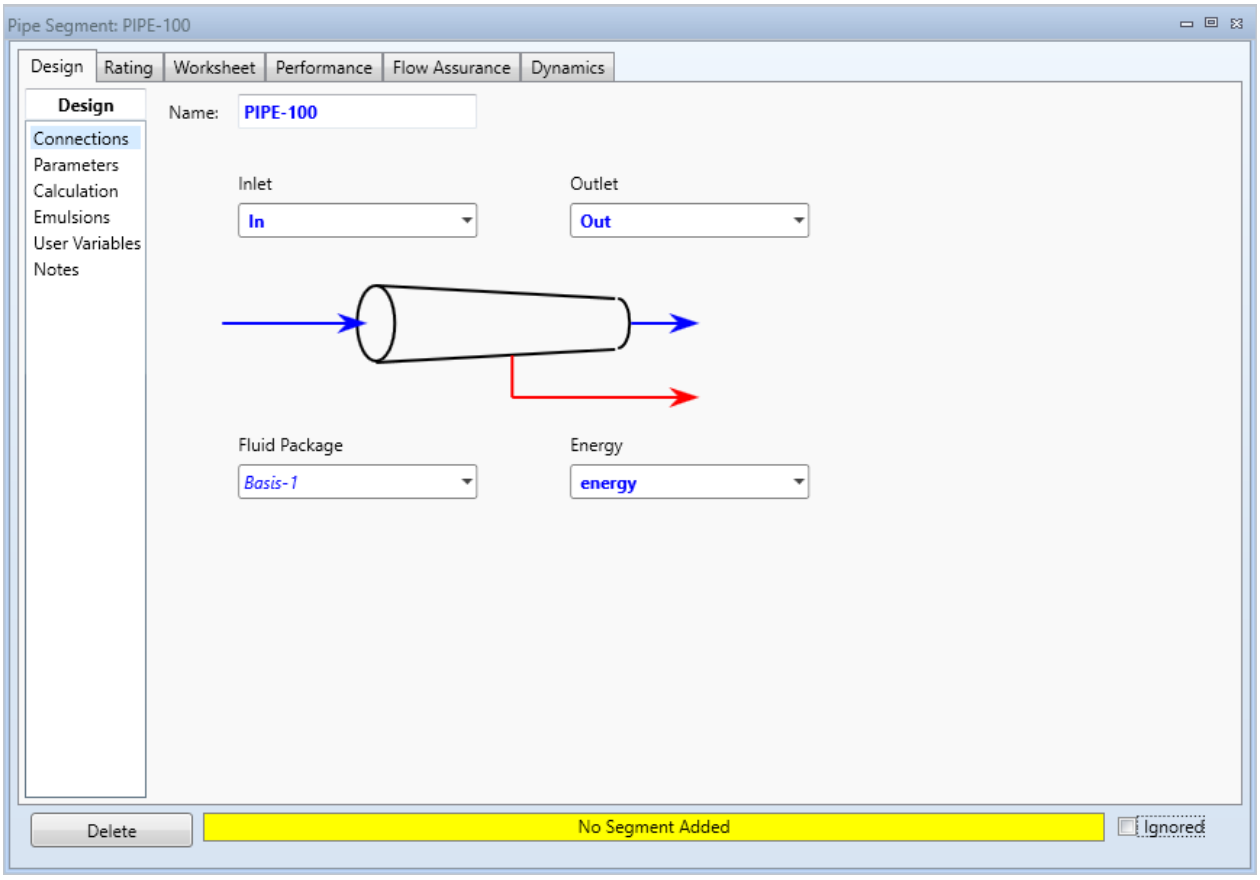
According to our given problem, we are supposed to use ASME steam package for our problem. So, from the dropdown menu, we will select the ASME steam option. Now our components and property databases are ready. We are ready to move to Simulation Tab by clicking  the button.



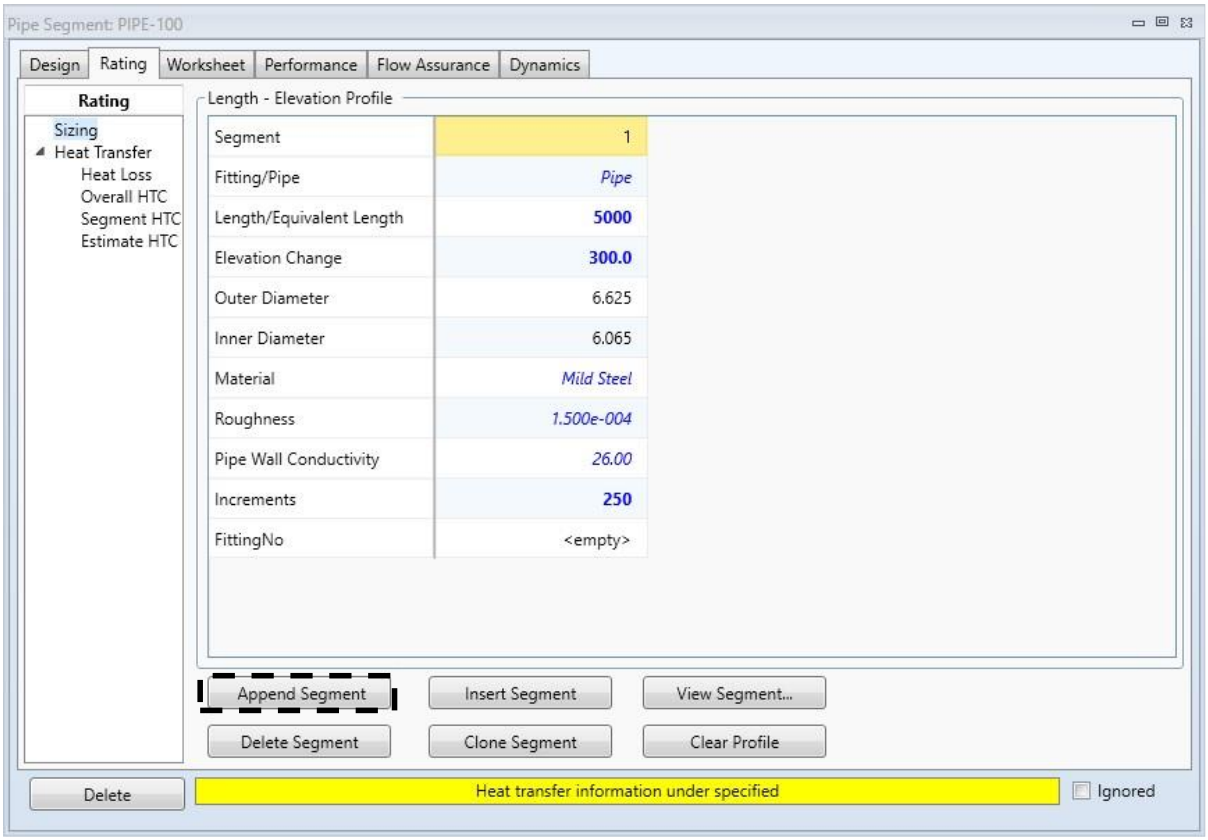
Step 7:

This is the most important step. First, we will drag  the 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, output, and energy stream names. The colour bar is yellow as we have not entered any segment yet.

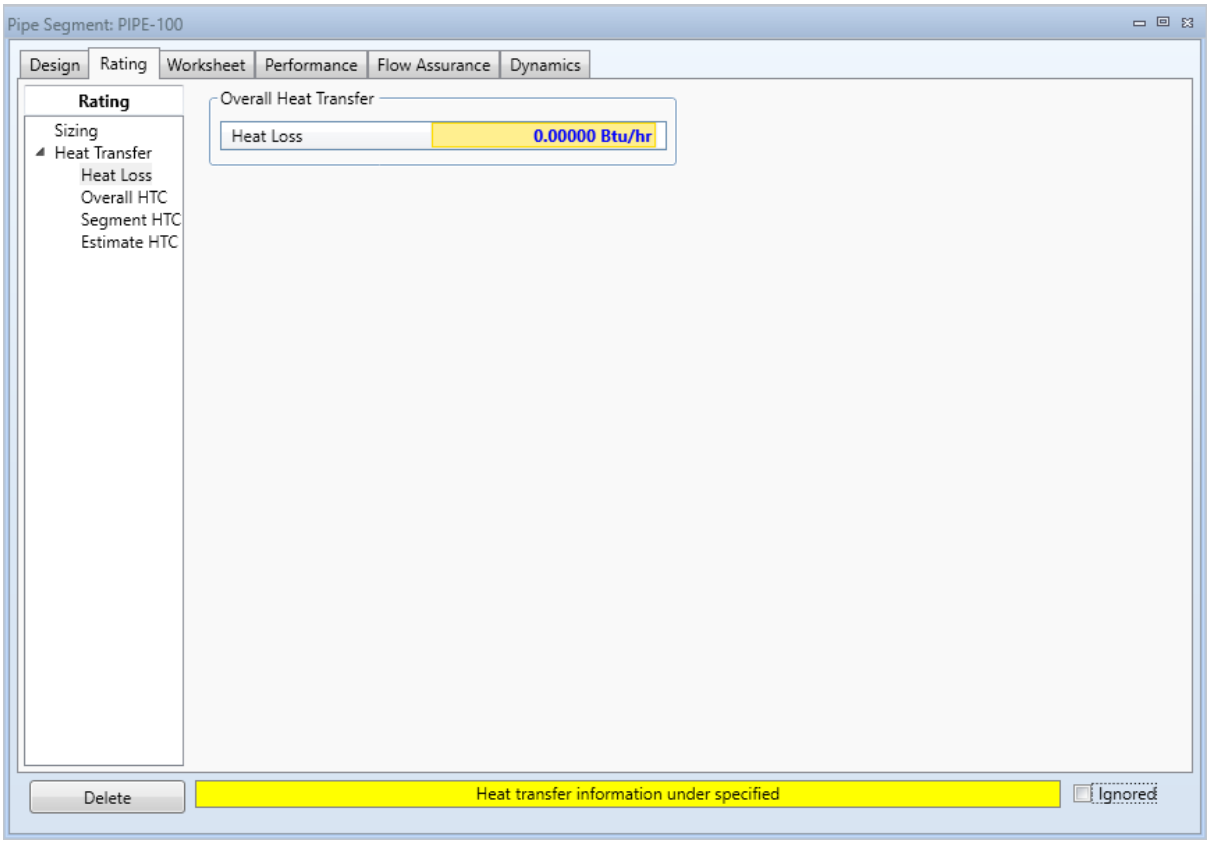




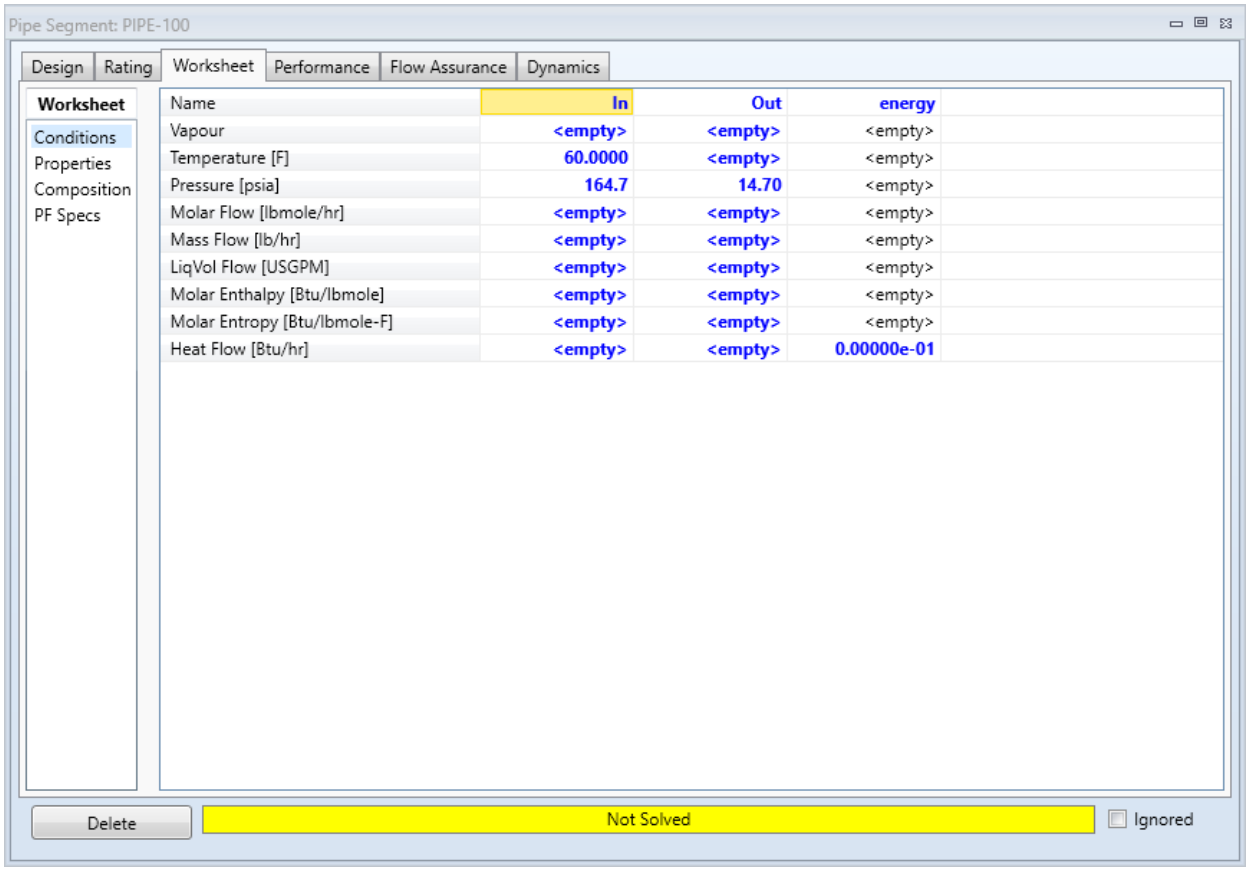
Now we will click on Rating tab which will open another dialogue where we can enter the inlet and outlet stream properties according to the given problem statements. Here we will click on the Append Segment icon to add new segment. Here we will enter all the data of the pipe.



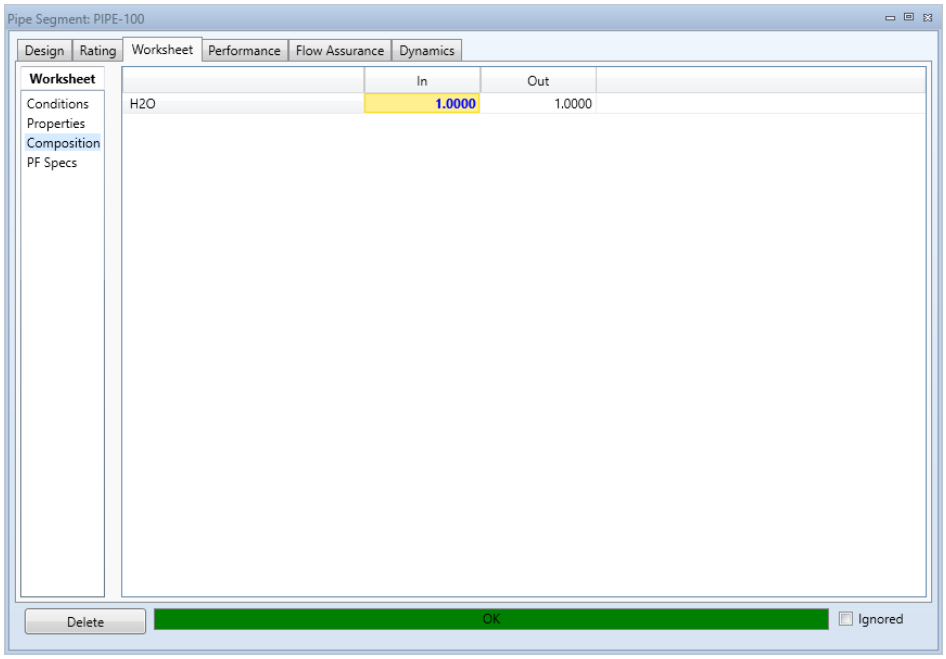
Now we will click on Heat Transfer Heat Loss tab which will open another dialogue where we can enter the inlet and outlet stream properties according to the given problem statements.



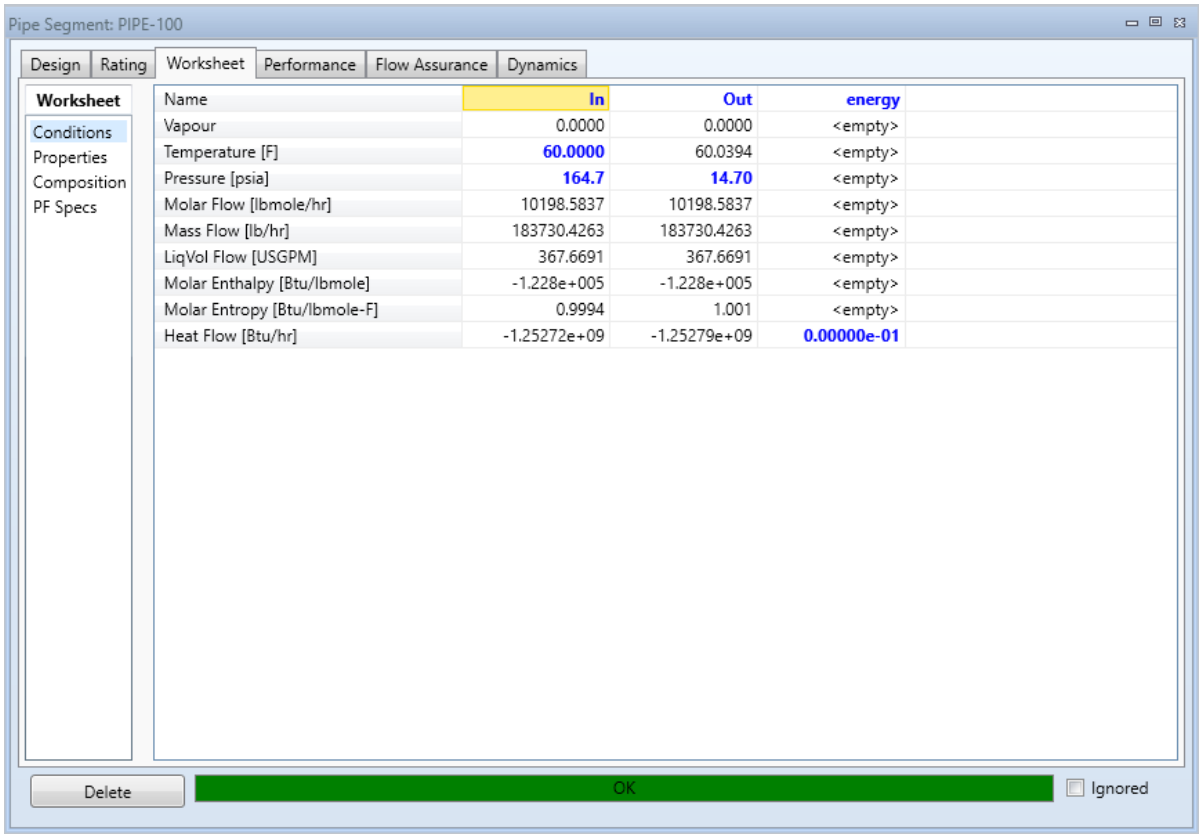
Now we will click on Worksheet tab which will open another dialogue where we can enter the inlet and outlet stream properties according to the given problem statements.



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 Composition tab which will open another window where we can enter the composition.



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



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