Process Simulation Using DWSIM:

A Free and Open-Source Chemical Process Simulator

By

Viraj Desai, Process Engineer

Email: Viraj Desai

Level: Basic

Version: 0.00

Aug 2022

PREFACE

The manual “Process Simulation Using DWSIM” presents a set of basic exercises using a free and open-source chemical process simulator “DWSIM” and can be utilized to establish process simulation laboratory as part of undergraduate chemical engineering degree or in allied degree curriculum. Simulation covers topics across the broad spectrum of chemical engineering courses covering mixing, reaction, phase equilibrium, heat, and mass transfer operations. The problem statements are rightly placed at the beginner’s level with each exercise completes in terms of sufficient instructions that enable the learner to perform the exercise with ease on their own. Supplementary self-learning exercises are also provided for simulation experiments to further aid a curious learner.

Thanks

Viraj Desai

P.E. O&G

Disclaimer

All the exercises are strictly restricted to learning only and not meant to be used in real world application.

PROCESS SIMULATION USING DWSIM: A FREE AND OPEN-SOURCE CHEMICAL PROCESS SIMULATOR

PREAMBLE

DWSIM is an open-source CAPE-OPEN compliant chemical process simulator. It features a Graphical User Interface (GUI), advanced thermodynamics calculations, reactions support and petroleum characterization / hypothetical component generation tools. DWSIM can simulate steady-state, vapor–liquid, vapor–liquid-liquid, solid–liquid and aqueous electrolyte equilibrium processes and has built-in thermodynamic models and unit operations (https://en.wikipedia.org/wiki/DWSIM). It is available for Windows, Linux and Mac OS.

The objective of the course is to create awareness of the open-source process simulator "DWSIM" among prospective graduates and practicing process engineers. The course will cover basic aspects of create flow sheet in DWSIM and simulation of simple units such as Mixer, Splitters, CSTR, Distillation column, Pumps, Turbines, Compressors, etc.

Target Audience

- Junior Interns in Process Firms
- III / Final year B. Tech. Chemical Engineering students
- M. Tech. Chemical Engineering students
- Practicing Process Engineers

Table of Contents

List of Figures

1 MIXING OF IDEAL LIQUID STREAMS

Objective

Develop a simple process sheet to mix two liquid streams and estimate the flow rate and composition of outlet stream.

Data

- Inlet stream 1: 10 mol % Methanol solutions flowing at 20 kmol/h
- Inlet stream 2: 80 mol % Methanol solutions flowing at 10 kmol/h
- Both the streams are at 30 °C and at 1 bar pressure
- The liquid streams can be considered as ideal

DWSIM Blocks Used

- Mixer
- Material Stream
- Indicators (Digital or Analog)

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - CLASSIC UI). Click on “New steady state Simulation” as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add components required to solve the problem statement. In the present case, add Water and Methanol. Ensure all components are added from the same property database. For instance, in this case, both components are added from “ChemSep” database.
3. Specify the thermodynamic package as Raoult’s law.
4. Customize the system of units for the simulation and click “Next”.
5. The flow sheeting section of simulation window will be opened. First, let provide input and output streams for the unit operation to be performed. Drag and drop two Material streams available at the right, in the object palette. Rename them stream as “Inlet-Stream-1”and “Inlet-Stream-2”. These serve as input streams.
6. On clicking the “Inlet-Stream-1” and “Inlet-Stream-2” stream, general information about the stream will be displayed on the left side of screen. Specify the feed compositions, temperature, and pressure for the inlet streams. Once credentials are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as “Outlet Stream”. This serves as output stream.
8. Below the Unit Operation tab on left, locate the Stream Mixer block. Drag and drop into the flow sheet. Rename it as “Mixer”.
9. No separate specification is required for the “Mixer” block.
10. Add digital and analog indicators for the material streams and mixer as shown in the figure below and give targeting properties.

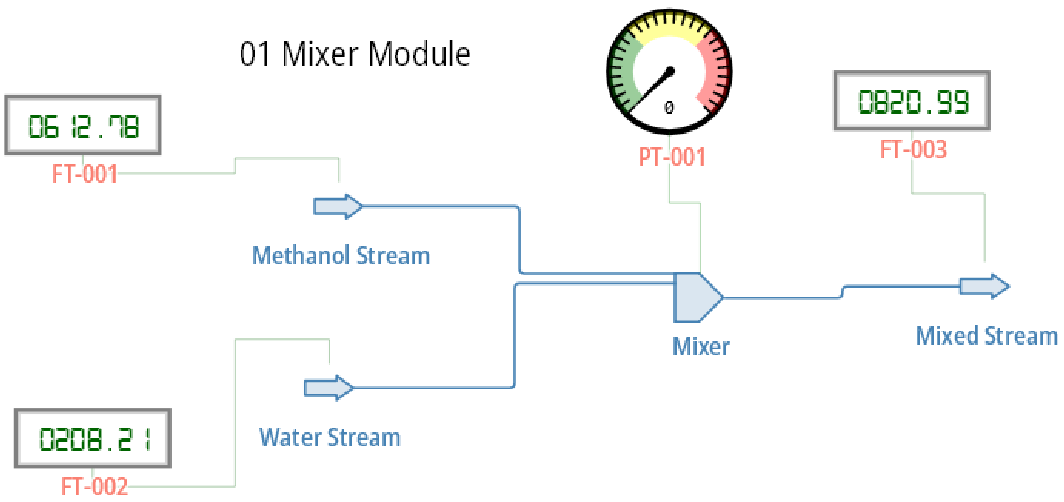


Figure 1 Mixer

11. Now, all necessary credentials required for simulation are added. It should be connected in a proper sequence. Click on “Mixer” block, the general information about the block is displayed on the right. Under “connections” tab, for all streams click the dropdown button and select the necessary connections. If all the connections are given correctly, all the blocks will turn blue.
12. Run the simulation by pressing “Solve flow sheet” button on the top corner of the screen.
13. To analyze/display the results, select on “Master property table” icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click “OK”. The property table will be opened showing all the results as shown in the figure below.

Properties	**Methanol Stream**	**Water Stream**	**Mixed Stream**
Temperature (K)	298.15	298.15	298.15
Pressure (Pa)	101325	101325	101325
Mass Flow (kg/s)	0.170218	0.057835	0.228053
Molar Flow (mol/s)	5.55556	2.77778	8.33334
Volumetric Flow (m3/s)	0.000211298	6.11026E-05	0.000269039
Density (Mixture) (kg/m3)	805.584	946.524	847.656
Molecular Weight (Mixture) (kg/kmol)	30.6392	20.8206	27.3663
Specific Enthalpy (Mixture) (kJ/kg)	-1256.07	-2053.47	-1458.29

Specific Entropy (Mixture) (kJ/[kg.K])	-4.08201	-6.72148	-4.652
Molar Enthalpy (Mixture) (kJ/kmol)	-38485	-42754.6	-39908.2
Molar Entropy (Mixture) (kJ/[kmol.K])	-125.07	-139.945	-127.308
Thermal Conductivity (Mixture) (W/ [m.K])	0.212675	0.422925	0.248449

2 Determination of Thermo-physical Properties of Pure Component

Objective

Determine the thermo-physical properties of pure component as function of temperature and pressure. For instance, determine the specific heat capacity of liquid water at 1 bar from 30 °C to 90 °C

Data

- Fluid: Water
- Thermodynamic model: Ideal

DWSIM Block

- Material Stream
- Digital indicators

Procedure

1. Start a new DWSIM simulation (DWSIM VER 8.0 - CLASSIC UI). Click on “New steady state simulation” as a template for new simulation
2. The simulation configuration window will be opened. The specification page will appear. Select the component required for the simulation. In this case Water is added.
3. Select and add the property package (Raoult’s law) and click “Next”. Add the default flash algorithm for the simulation. Click “Next”.
4. Choose the desired system of units for the simulation and click “Next”.
5. The flow sheeting section will be opened.

6. Add a material stream and give the conditions provided in the data and add digital indicators for targeting the properties

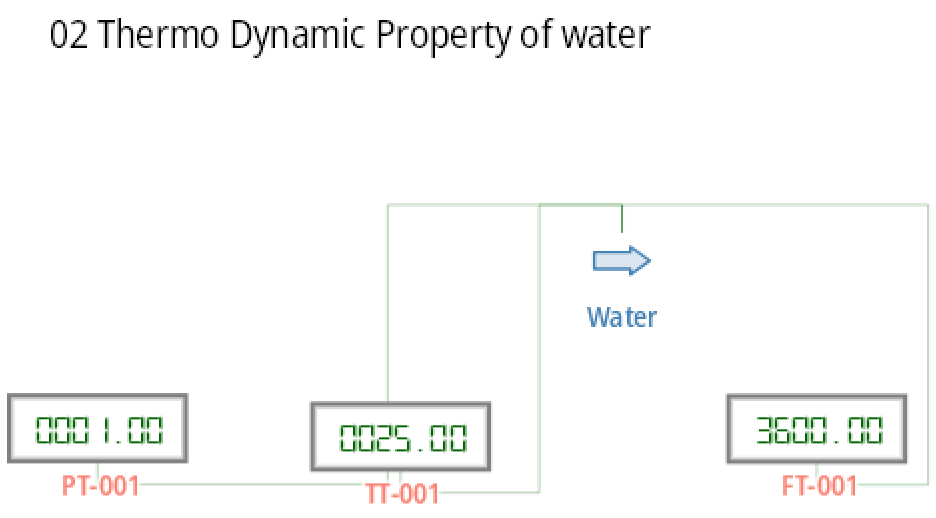


Figure 2 Thermodynamic property of water

7. In the top menu, under the “Tools” tab, click on “Pure component property viewer” to generate thermo-physical property data.

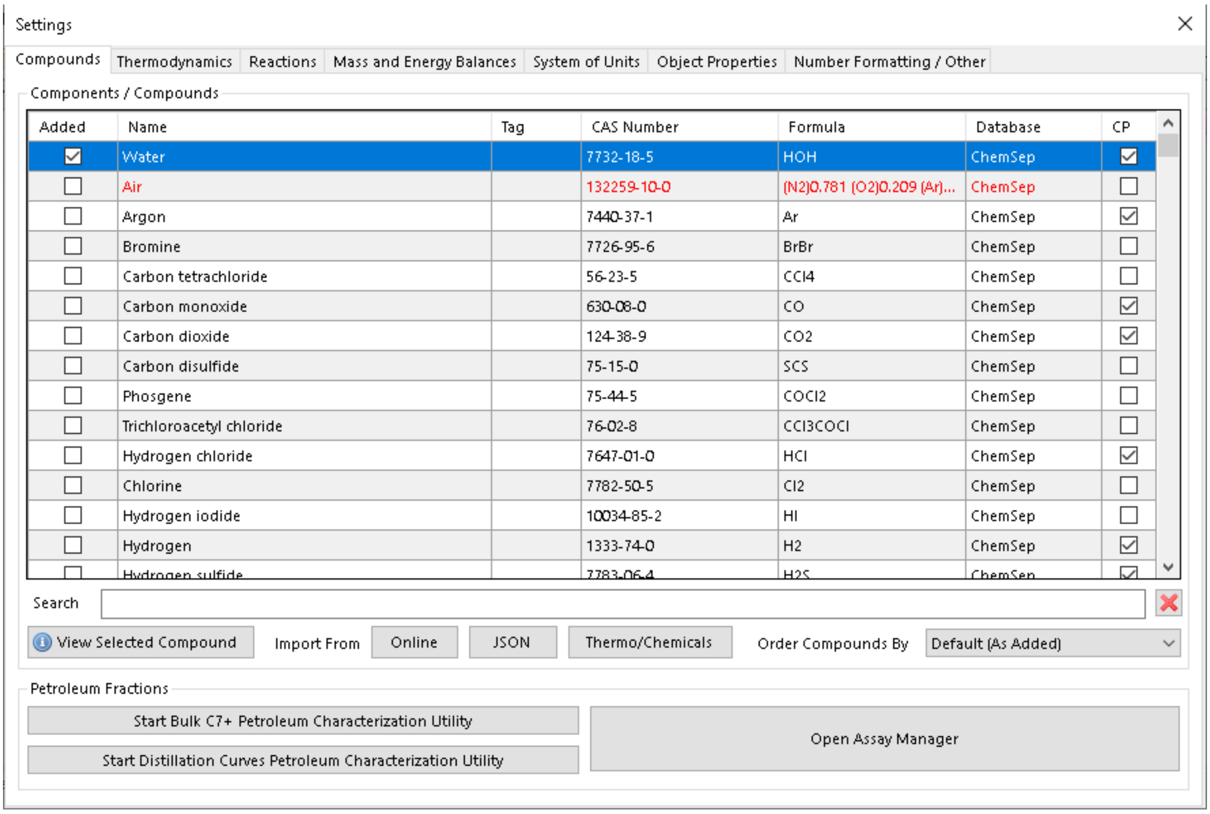


Figure 3 Pure Component property viewer

8. On clicking the tab, pure component property window will appear. Select the component for which the thermo-physical data must be generated. In this case it is water.
9. Once you selected the component, the appearance of window will be changed as shown in figure.

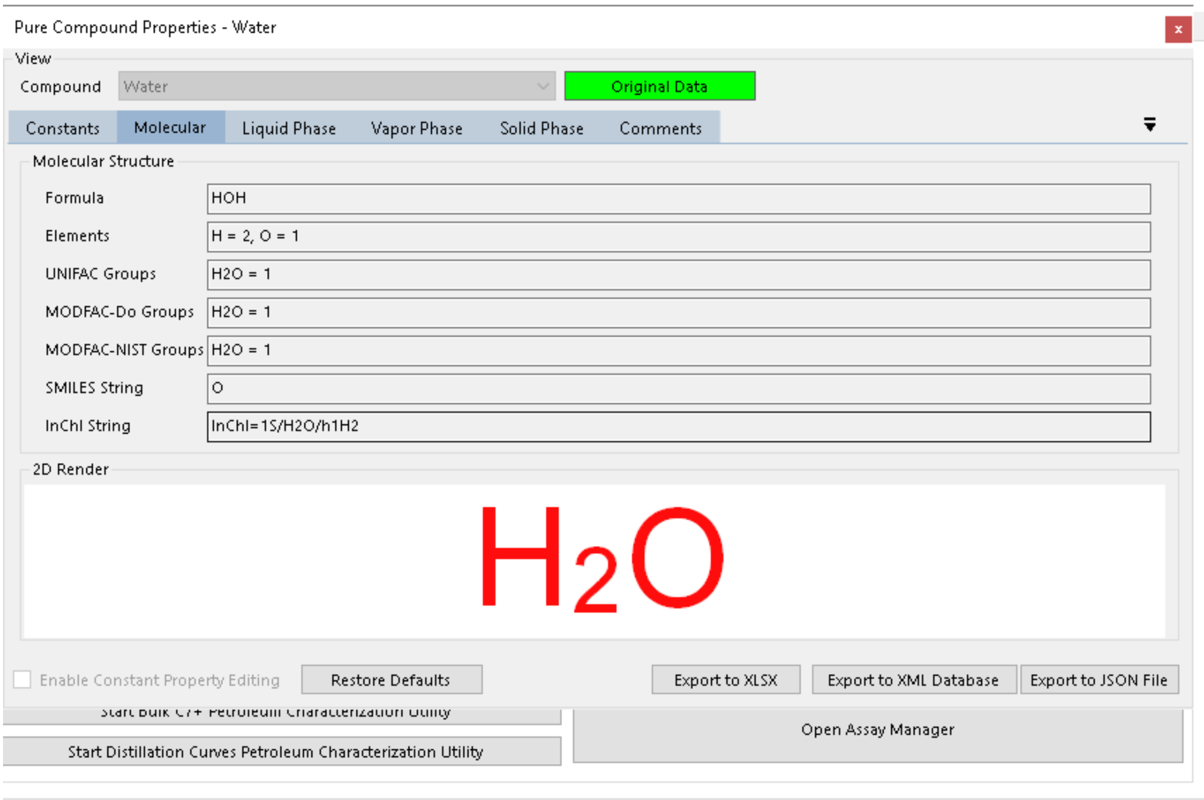


Figure 4 Pure Component Property viewer

10. Below the component tab, identify a tab named "Liquid-phase". On clicking this tab, a graph for temperature versus specific heat capacity of water will be generated.

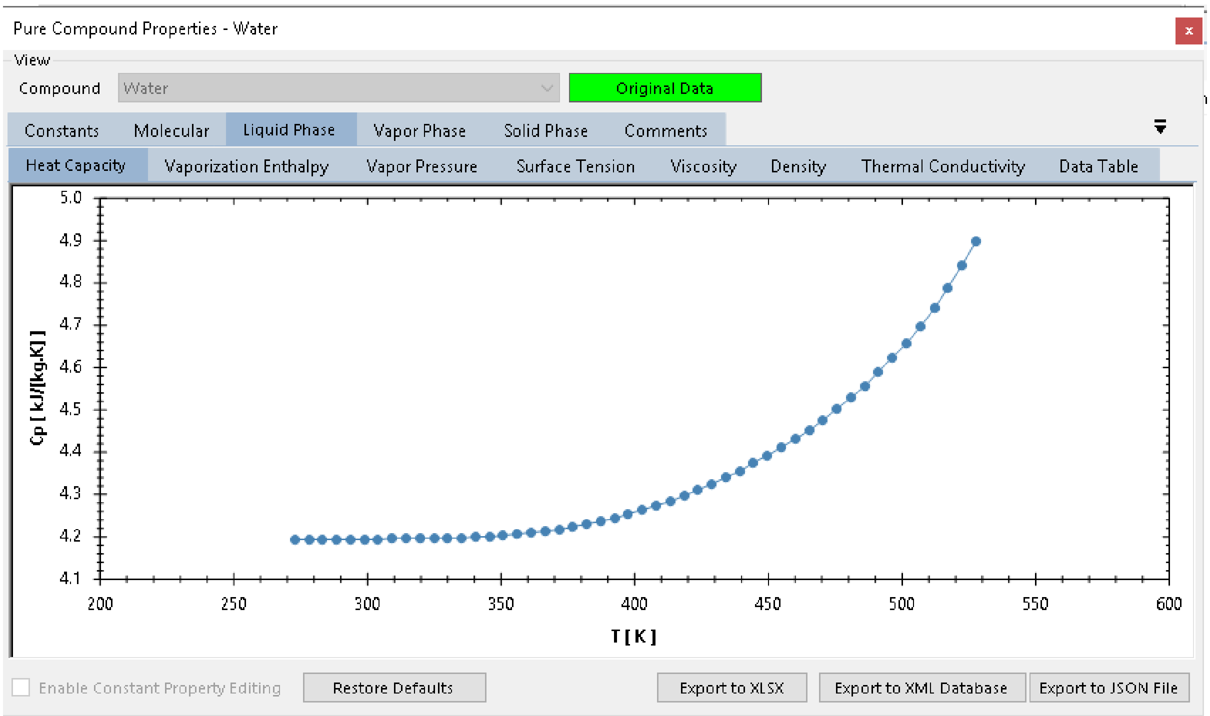


Figure 5 Heat Capacity of water

11. Similarly you can also generate for vapourization enthalphym vapour pressure, surface tension, viscosity,density and so on.
12. To view the results in tabulated form, click on "Data-table" option where, all the thermo-physical properties are tabulated.

3 GENERATION OF VLE DATA OF BINARY COMPONENT SYSTEM

Objective

Generate vapour-liquid equilibrium data (VLE) for a binary component system

Data

- Fluid components: Benzene - Toluene
- Pressure = 1 atm
- Thermodynamic model: Ideal

DWSIM Block

- Material Stream

Procedure

1. Start a new DWSIM simulation (DWSIM VER 8.0 - CLASSIC UI). Click on "New steady state simulation" as a template for new simulation
2. The simulation configuration window will be opened. The specification page will appear. Select the components required for the simulation, namely "Benzene" and "Toluene". Ensure, components are added from same property database. In this illustration, both components are added from "ChemSep" database.
3. Select and add the property package (Raoult's law) and click "Next". Add the default flash algorithm for the simulation. Click "Next".
4. Choose the desired system of units for the simulation and click "Next".
5. The flowsheeting section will be opened.
6. Add a material stream and specify the data.

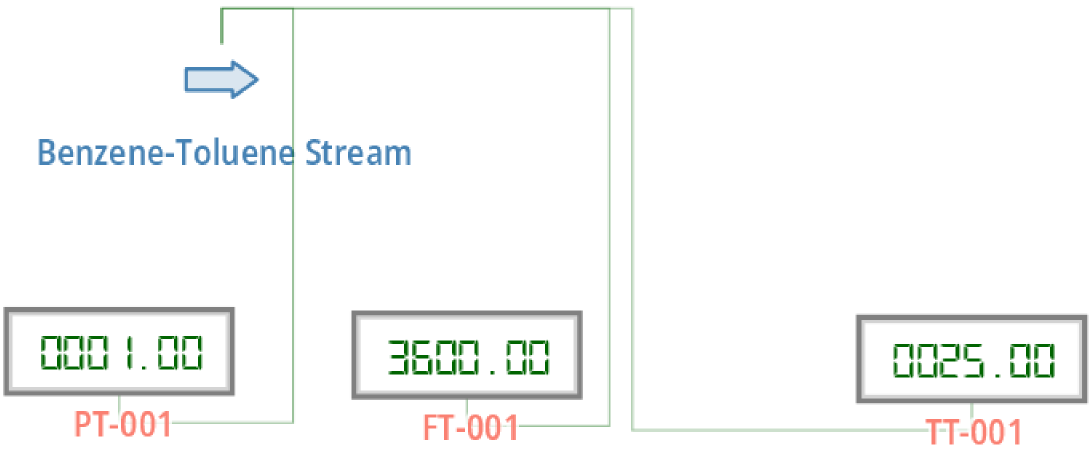


Figure 6 Benzene-Toluene Stream

7. To generate binary VLE, at least one material stream is required in the flowsheeting section. Hence, click on material stream object at the object palette and drag it to the flowsheet section.
8. Click the "material stream" to open its specification window. Next to the object name, you will find an icon, click it to attach utility and under utility add an "Binary Envelope" as shown in the figure below.

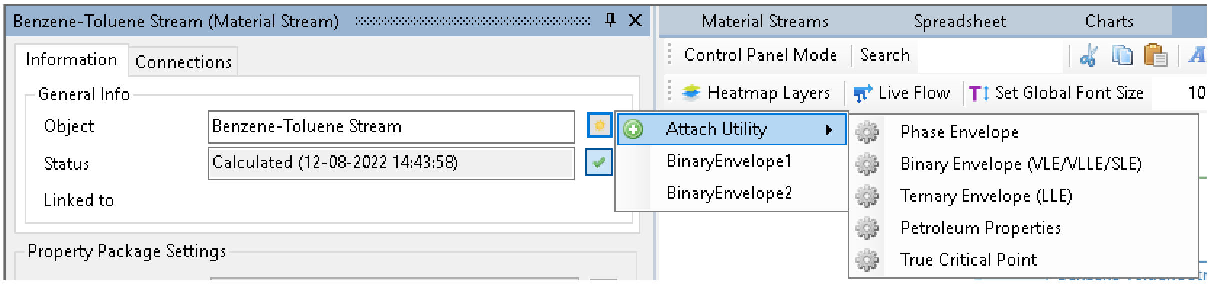


Figure 7 Add utility Window

9. A new window opens. Enlarge it. Use the different pull down menus available in the window to generate the Txy and VLE plot for Benzene and Toluene at given pressure. Provide settings as shown below and click at "Calculate" to see the Txy plot.

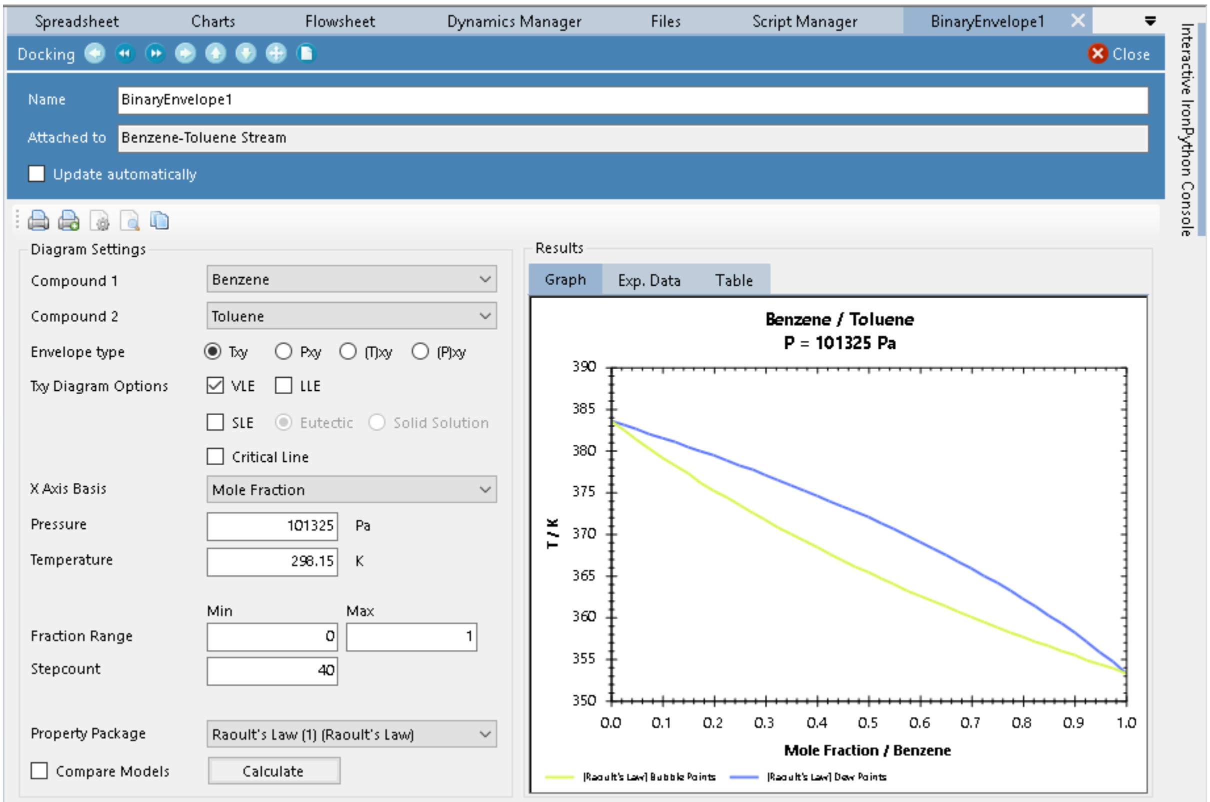


Figure 8 Binary Envelope Window T vs XY

10. Similarly click on Pxy and then on "Calculate" to generate Pxy plot.

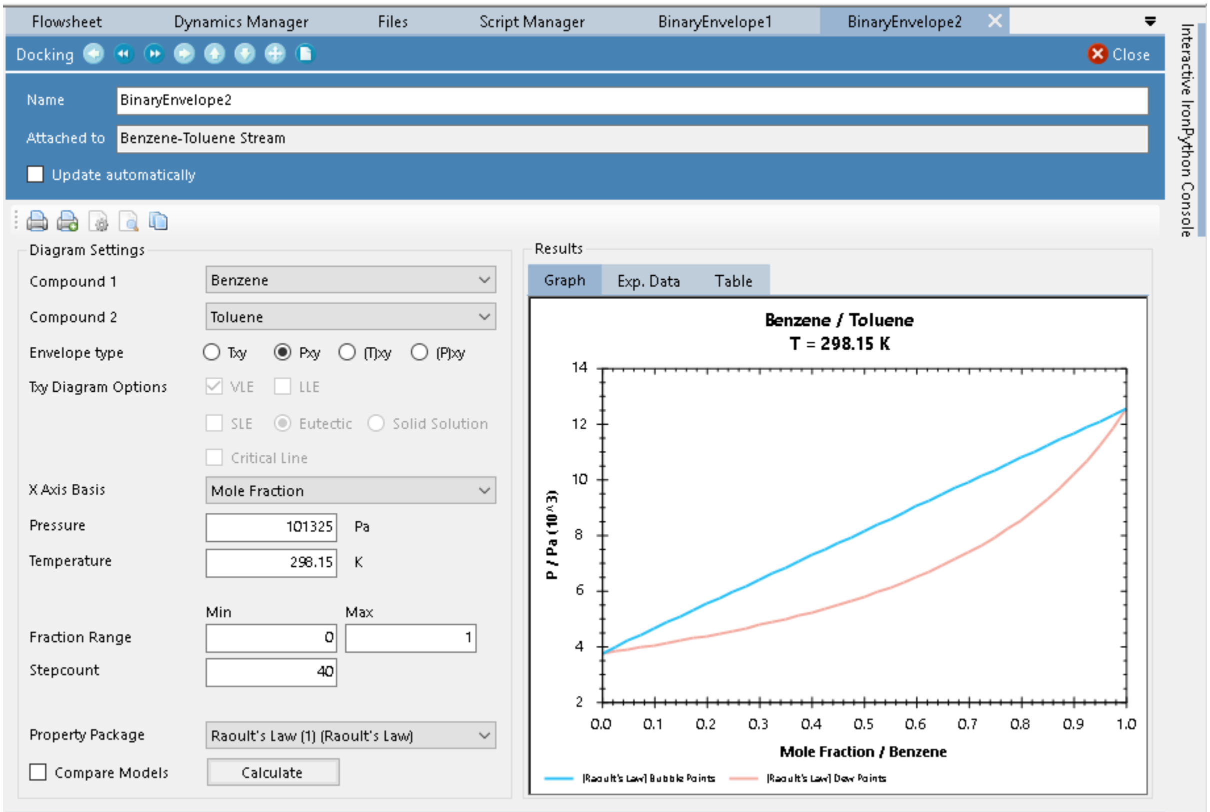


Figure 9 Binary Envelope Window P vs XY

11. Click on {T}xy and "Calculate" to generate {T}xy plot.

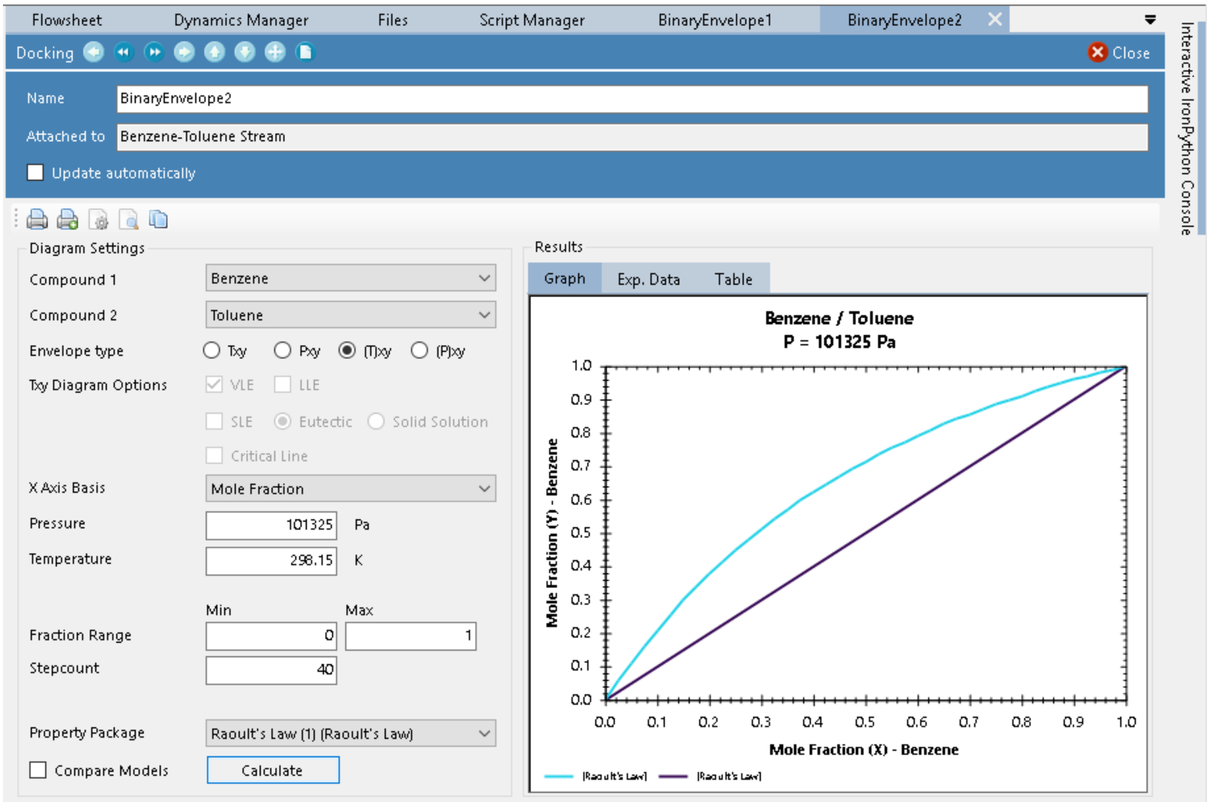


Figure 10 Binary Envelope Window X vs Y

4 SIMULATION OF A FLASH COLUMN

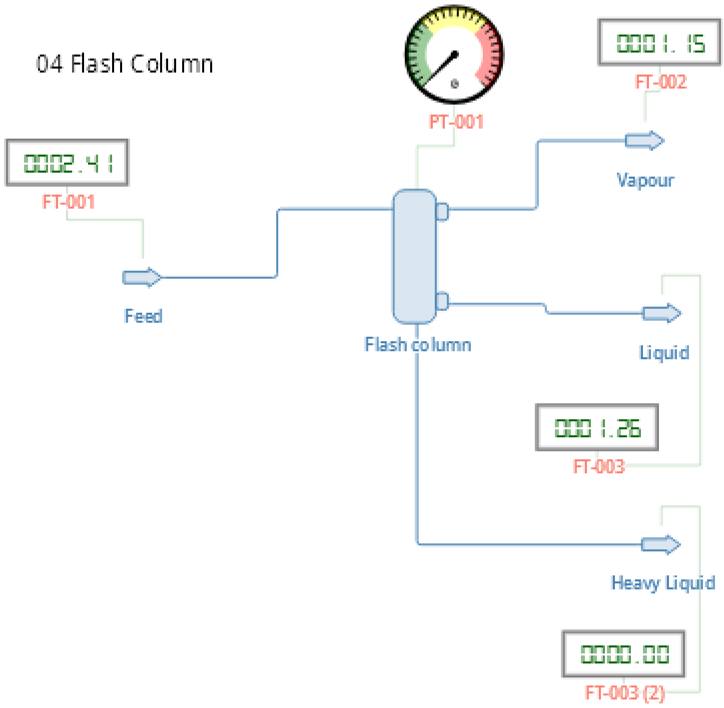
Objective
Develop a simple process flow sheet to estimate the liquid and vapour composition of multi-component mixture undergoing partial vaporization.

Data
- Components: n-pentane, n-hexane, and n-heptane
- Feed composition: 25 mol % n-pentane, 45 mol% n-hexane, and 30 mol% n-heptane
- Basis: 100 kmol/h
- Operating conditions
- Temperature = 69 °C
- Pressure = 1.013 bar

DWSIM Blocks Used
- Gas-Liquid Separator

Procedure
1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on “New steady state Simulation” as a template for new simulation.
2. The simulation configuration window will be opened. It shows a specification page. Add the three components - n-pentane, n-hexane, and n-heptane. Ensure that all the components are added from same property package. Example: All the 3 components are selected from Chemsep database. Click “Next” button.
3. Specify the thermodynamic package as Raoult’s law.
4. Customize the system of units for the present simulation and click “Next”.
5. The flowsheeting section of simulation window will be opened. It is necessary to provide input and output streams for the unit operation to be performed. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as “Feed-In”. This serves as the input stream.
6. On clicking the “Feed-In” stream, general information about the stream will be displayed on the left side of screen. Specify the feed compositions, flow rate, temperature, and pressure

for the stream, once composition and flow rate are specified for the inlet stream, the color



of stream turns blue.

Figure 11 Flash Column

7. Add two more Material streams i.e., Drag and drop it into the flowsheet. Rename those streams as “Vap-Out”, “Liq-Out” & “Heavy-Out”. These serves as output streams.
8. Below the unit operation tab on left, locate the Gas-Liquid Separator block. Drag and drop into the flow sheet. Rename it as “Flash Column”.
9. Now, all necessary credentials required for simulation are added. It should be connected in a proper sequence. Click on “Flash Column” block, the general information about the block is displayed on the right. Under “connections” tab, for all streams click the dropdown button and select the necessary connections. If all the connections are given correctly, all the blocks will turn blue.
10. Add Flow transmitters as indicators in all the streams and target mass flow as property.
11. For column add a pressure transmitter, and target pressure.
12. Run the simulation by pressing “Solve flow sheet” button on the top corner of the screen. It will be in the shape of Triangle.
13. To analyze/display the results, select on “Master property table” icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which must be shown in output and click “OK”. The property table will be opened showing all the results as shown in the figure below.

	Heavy Liquid	**Feed**	**Vapour**	**Liquid**
Temperature (K)	343.15	343.15	343.15	343.15
Pressure (Pa)	101300	101300	101300	101300
Mass Flow (kg/s)	0	2.41324	1.15056	1.26268
Molar Flow (mol/s)	0	27.7778	13.8003	13.9775

Volumetric Flow (m3/s)	0	0.390702	0.388664	0.0020377
Density (Mixture) (kg/m3)	∞	6.17669	2.9603	619.662
Molecular Weight (Mixture) (kg/kmol)	0	86.8767	83.372	90.3371
Specific Enthalpy (Mixture) (kJ/kg)	0	-95.2365	79.2048	-254.188
Specific Entropy (Mixture) (kJ/[kg.K])	0	-0.162612	0.349025	-0.628817
Molar Enthalpy (Mixture) (kJ/kmol)	0	-8273.83	6603.46	-22962.6
Molar Entropy (Mixture) (kJ/[kmol.K])	0	-14.1272	29.0989	-56.8055
Thermal Conductivity (Mixture) (W/[m.K])	0	0.0624136	0.0178495	0.106413

5 SIMULATION OF A CSTR FOR LIQUID PHASE REACTION

Objective

Develop a simple process sheet to determine the exit composition from a Continous stirred tank reactor (CSTR)

Data

- Reaction: Ethylene glycol production in CSTR: Ethylene oxide reacts with water to form Ethylene Glycol
  $C_2H_4O_{(l)} + H_2O_{(l)} \rightarrow C_2H_6O_{2(l)}$
- Inlet stream: 26 m^3/h of aqueous solution of Ethylene oxide with a mol fraction of 15% and rest water. Reaction carried out at 55 °C under atmospheric condition in CSTR of 2 m^3
- Reaction rate: $-r = kC_{E0}$
- Where C_{EO}: Molarity of Ethylene oxide; k: rate constant = 0.005 1/s

DWSIM Blocks Used

- Continuous Stirred Tank Reactor (CSTR)

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - CLASSIC UI). Click on “New steady state Simulation” as a template for new simulation
2. The simulation configuration window will be opened. Add 3 components for the simulation - Ethylene oxide, Ethylene Glycol and Water
3. Ensure that all the components are added from same property package. Example: All the 3 components are selected from Chemsep database. Click “Next” button.
4. Select and add the property package and click “Next”. Add the default flash algorithm for the simulation. Click “Next”.
5. The flowsheeting section of simulation window will be opened. Drag and drop the Material stream from the object palette and rename the stream as “Feed”. This serves as input.
6. On clicking the “Feed” block, general information about the block will be displayed on the left of the screen.
7. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
8. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename it as “Product”. This serves as output stream.
9. Add an energy stream which is available in the object palette below the Material stream.
10. Below the Unit Operation tab, locate the “Continous Stirred Tank Reactor” CSTR block. Drag and drop into the flow sheet. Rename it as “CSTR”.
11. Under “Tools” tab in select “Reaction Manager” tab. choose the type of reaction i.e., “Equilibrium reaction” for this problem. A dialogue box will appear. Give an appropriate

name and description about the reaction.

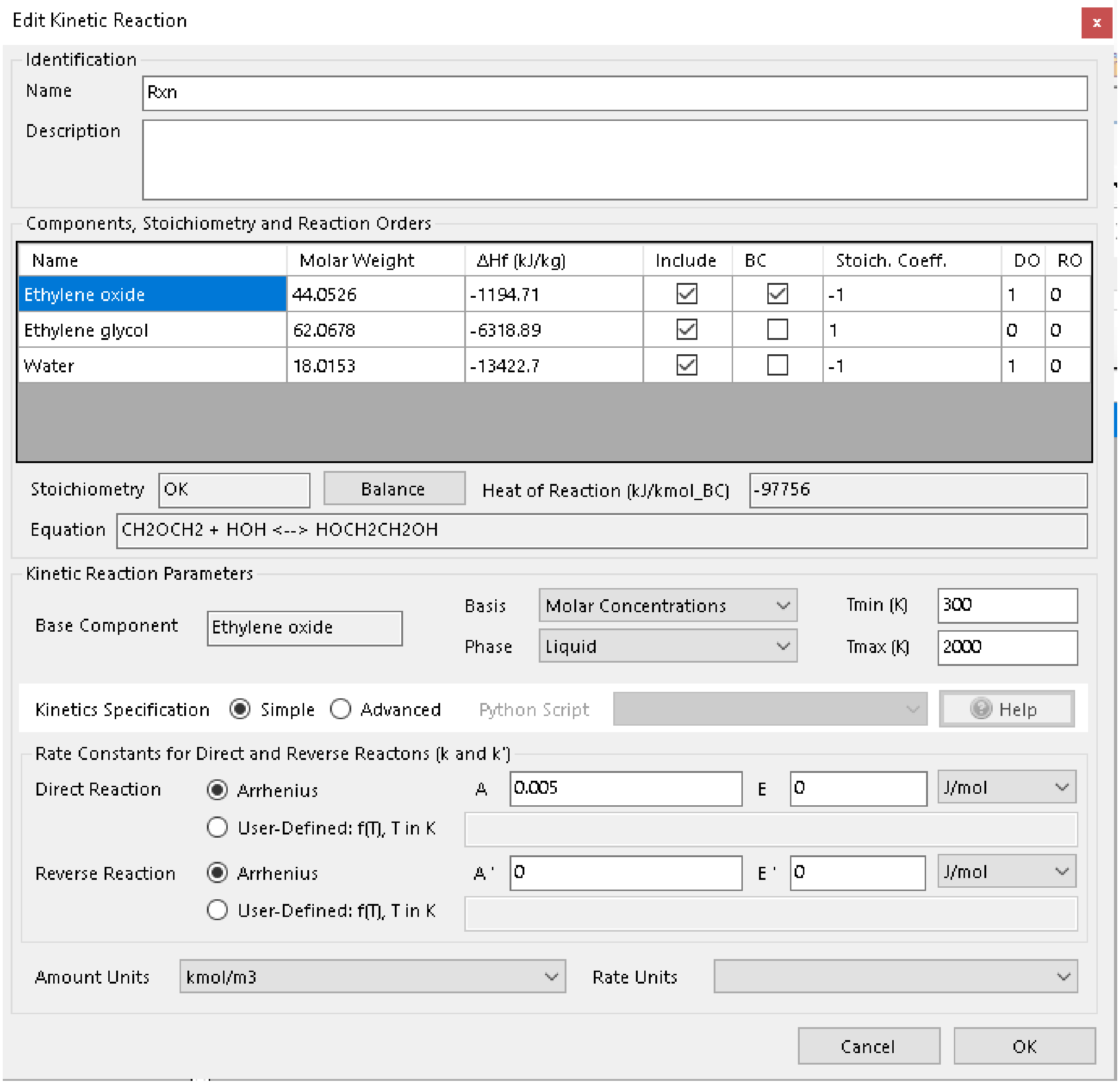


Figure 12 Reaction window

12. Select the checkboxes adjacent to the component names, which has to be included in the reaction. Give the stoichiometry of the reaction and choose appropriate base component.
13. In this case the base component is "Ethylene Oxide". Once base component, stoichiometry is specified, a text "OK" appears in the stoichiometry tab. Specify the rate constant of the reaction.
14. By default, the basis is activity and liquid phase. In this case it has to be changed to Molar concentration and specify the phase as liquid.
15. Specify the rate constant of the reaction and click "OK".
16. Click on "CSTR-REC" block, the general information about the block is displayed on the right. Under "connections" tab, for all streams click the dropdown button and select the necessary connections. If all the connections are given correctly, the blocks will turn blue.

17. Add 2 analyzers from the indicators panel and specify molar fraction (liquid 1) / ethylene glycol as the targeting property for both feed and product stream.

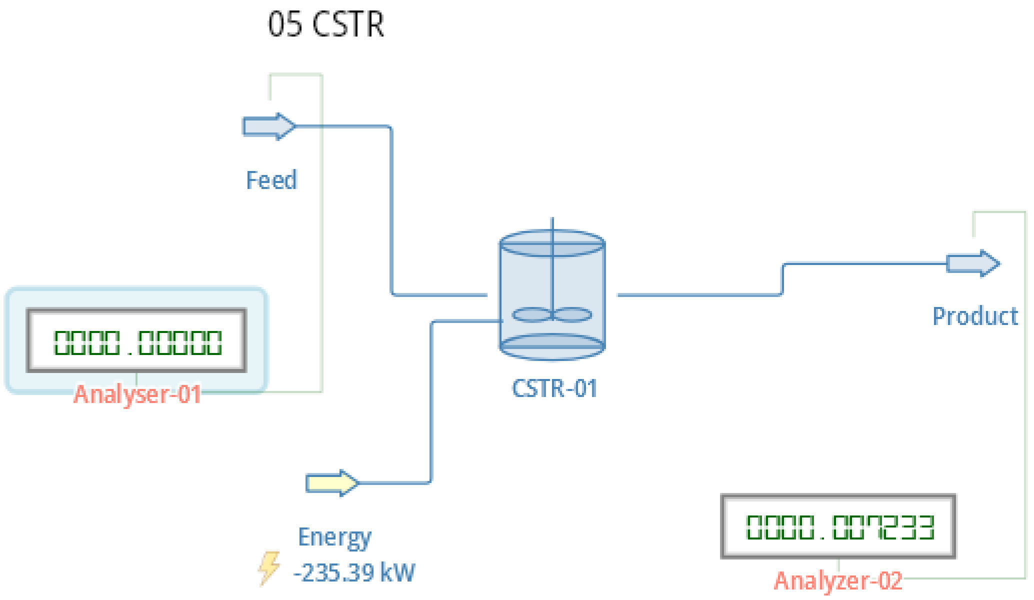


Figure 13 CSTR

18. Run the simulation by pressing "Solve flow sheet" button on the top corner of the screen.
19. To analyze/display the results, select on "Master property table" icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click "OK". The property table will be opened showing all the results as shown in the figure below.

	Feed	**Product**
Temperature (K)	328.15	328.15
Pressure (Pa)	101325	101325
Mass Flow (kg/s)	6.91362	6.91362
Molar Flow (mol/s)	315.39	313.125
Volumetric Flow (m3/s)	0.00722222	0.00720902
Density (Mixture) (kg/m3)	957.271	959.023
Molecular Weight (Mixture) (kg/kmol)	21.9209	22.0794
Specific Enthalpy (Mixture) (kJ/kg)	-1795.78	-1797.8
Specific Entropy (Mixture) (kJ/[kg.K])	-4.93172	-4.90951
Molar Enthalpy (Mixture) (kJ/kmol)	-39365	-39694.4
Molar Entropy (Mixture) (kJ/[kmol.K])	-108.108	-108.399
Thermal Conductivity (Mixture) (W/[m.K])	0.423148	0.424053
Molar Fraction (Mixture) / Ethylene glycol	0	0.00723287
Mass Fraction (Mixture) / Ethylene glycol	0	0.0203324

6 SIMULATION OF A DISTILLATION COLUMN

Objective

Develop a simple process flow sheet to estimate distillate and bottom composition of a distillation column

Problem statement

100 kmol/h of an equimolar mixture of benzene and toluene at 70°C and 1 atm pressure is to be separated by staged distillation column. A reflux ratio of 3 is used. Composition of benzene in the distillate should be 99% (by mol) toluene in the bottom should be 99% (mol). A total condenser and reboiler, both at 1 atm pressure are used. Determine the actual no. of stages, minimum reflux ratio and the minimum no. of stages for the operation.

DWSIM Blocks Used

- Shortcut Column
- Material Streams
- Energy Streams
- Indicators

Thermodynamic Package

- UNIFAC.

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - CLASSIC UI). Click on “New steady state Simulation” as a template for new simulation
2. The simulation configuration window will be opened. Add the two components required for simulation - Benzene and Toluene. Ensure that all the components are added from same property package. Example: All the 2 components are selected from Chemsep database. Click “Next” button.
3. Select and add the property package and click “Next”. Add the default flash algorithm for the simulation. Click “Next”.
4. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream from the object palette. Rename it as “Feed”. This serves as input stream.
5. On clicking the “Feed” stream, general information about the block will be displayed on the left of the screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams. Once composition and flow rate are specified for the inlet streams, the color of stream turns blue. Specification for the inlet stream
6. Add two more Material streams i.e., Drag and drop them into the flow sheet. Rename them as “Distillate” and “Bottoms”. These serves as output streams.
7. Add two energy streams, one is for condenser duty (C-Duty) and the other is for re-boiler duty (R-Duty).

8. Below the Unit Operation tab, locate the “Shortcut Column” block. Drag and drop into the flow sheet. Rename it as “DC”.

Figure 14 Columns Window

9. Click on “DC” block, the general information about the block is displayed on the left of the screen. Provide calculation parameters as shown in the screenshot given below
10. Under Column configuration select “connections” tab. Click the dropdown button and give appropriate connections. If all the connections are given correctly, the blocks will turn blue.
11. Add indicators on column, feed, distillate and bottom streams and target respective properties.
12. Run the simulation by pressing “Solve flow sheet” button on the top corner of the screen.

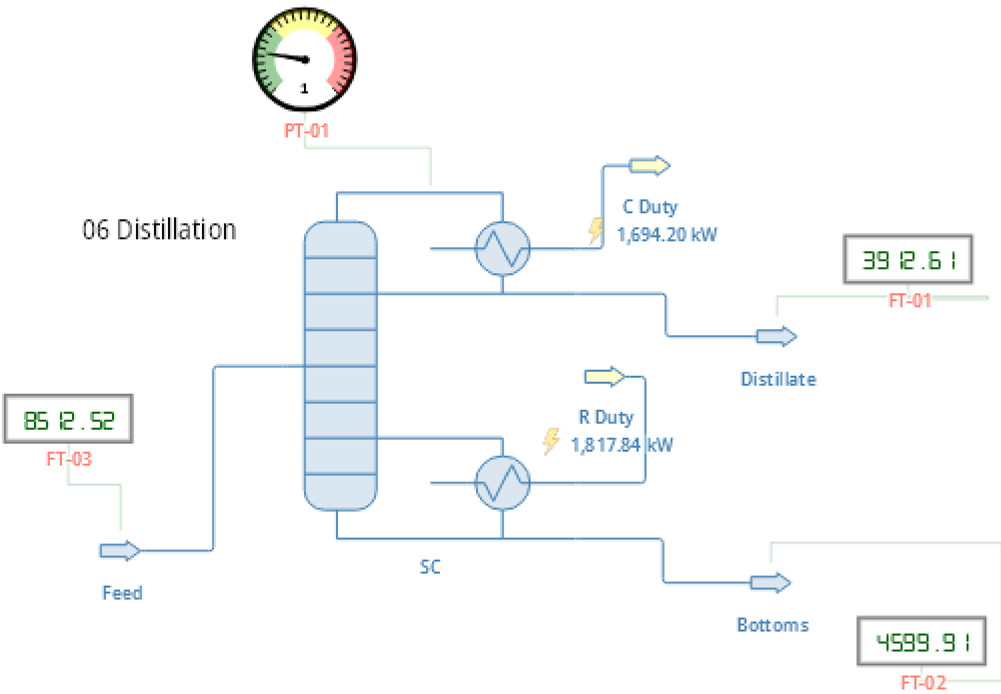


Figure 15 Distillation

13. To analyze/display the results, select on “Master property table” icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click “OK”. The property table will be opened showing all the results as shown in the figure below.

Property	**Value**	**Units**
Minimum Reflux Ratio	1.11363	
Minimum Number of Stages	9.15433	

Actual Number of Stages	12.7253	
Optimal Feed Stage	6.36267	
Stripping Liquid	69.4445	mol/s
Rectify Liquid	41.6667	mol/s
Stripping Vapor	55.5556	mol/s
Rectify Vapor	55.5556	mol/s
Condenser Duty	1694.2	kW
Reboiler Duty	13484.3	kW

Property table

	Bottoms	**Distillate**	**Feed**
Temperature (K)	1000.09	353.522	343.15
Pressure (Pa)	1.02668E+10	101325	101325
Mass Flow (kg/s)	1.27775	1.08684	2.36459
Molar Flow (mol/s)	13.8889	13.8889	27.7778
Volumetric Flow (m3/s)	1.12482E-05	0.0016119	0.00287003
Density (Mixture) (kg/m3)	113597	674.255	823.891
Molecular Weight (Mixture) (kg/kmol)	91.9982	78.2521	88.0251
Specific Enthalpy (Mixture) (kJ/kg)	8876.21	-328.694	-340.765
Specific Entropy (Mixture) (kJ/[kg.K])	0.00688362	-0.905619	-0.831621
Molar Enthalpy (Mixture) (kJ/kmol)	816595	-25721	-29007.7
Molar Entropy (Mixture) (kJ/[kmol.K])	0.63328	-70.8666	-70.7919
Thermal Conductivity (Mixture) (W/[m.K])	0.100323	0.126008	0.124969

7 DETERMINATION OF HEAT DUTY OF HEATER

Objective

Develop a simple process flow sheet to determine the heat duty required to heat a fluid to a desired temperature

Data

- Fluid: Water
- Inlet mass flow rate: 50 kg/h
- Inlet temperature: 25 °C
- Outlet temperature: 90 °C
- Pressure: 1 bar

DWSIM Blocks Used

- Heater
- Material stream
- Energy Stream
- Indicator

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on “New steady state Simulation” as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component “Water”
3. Specify the thermodynamic package as Raoult’s law.
4. Select the system of units for the simulation and click “Next”.
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as “Flow-In”. This serves as input stream.
6. Double click the “Flow-In” stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as “Flow-Out”. This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.

9. Below the Unit Operation tab on left, locate the Heater block. Drag and drop into the flow sheet. Rename it as "HEATER".

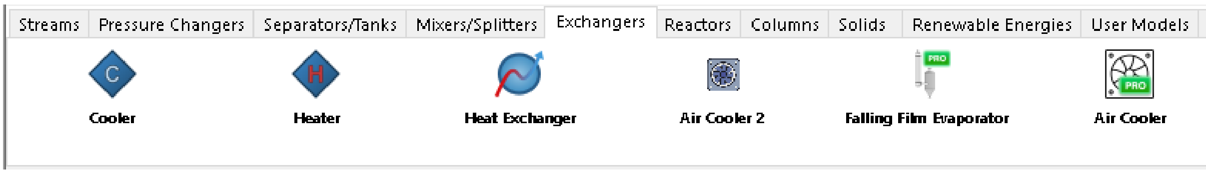


Figure 16 Exchangers Window

10. Click on "HEATER" block, the general information about the block is displayed on the right. Under "connections" tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose "Outlet Temperature". If all the connections are given correctly, all the blocks will turn blue.
11. Add temperature indicators on input and output stream and target the desired properties.
12. Run the simulation by clicking on "Solve flow sheet" icon / button on the top corner of the screen.

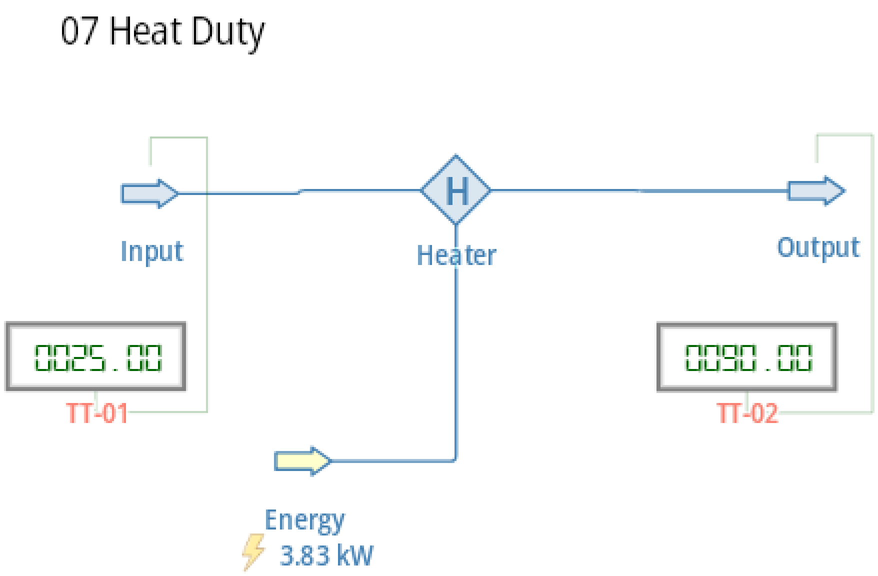


Figure 17 Heat Duty

13. To analyze/display the results, select on "Master property table" icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click "OK". The property table will be opened showing all the results as shown in the figure below.

	Input	**Output**
Temperature (K)	298.15	363.15
Pressure (Pa)	101325	101325
Mass Flow (kg/s)	0.0138889	0.0138889
Molar Flow (mol/s)	0.770951	0.770951
Volumetric Flow (m3/s)	1.39401E-05	1.43909E-05
Density (Mixture) (kg/m3)	996.327	965.118

Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	-2440.95	-2165.09
Specific Entropy (Mixture) (kJ/[kg.K])	-8.18698	-5.92785
Molar Enthalpy (Mixture) (kJ/kmol)	-43974.4	-39004.7
Molar Entropy (Mixture) (kJ/[kmol.K])	-147.491	-106.792
Thermal Conductivity (Mixture) (W/[m.K])	0.610248	0.676026

8 DETERMINATION OF POWER FOR A PUMP

Objective

Develop a simple process flow sheet to determine the power required to pressurize a fluid for a pump.

Data

- Fluid: Water
- Inlet mass flow rate: 50 kg/h
- Inlet Pressure: 2 bar
- Outlet Pressure: 5 bar
- Temperature: 50 °C

DWSIM Blocks Used

- Pump
- Material stream
- Energy Stream
- Indicators

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on "New steady state Simulation" as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component "Water"
3. Specify the thermodynamic package as Raoult's law.
4. Select the system of units for the simulation and click "Next".
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as "Flow-In". This serves as input stream.
6. Double click the "Flow-In" stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as "Flow-Out". This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.
9. Below the Unit Operation tab on left, locate thePump block. Drag and drop into the flow sheet. Rename it as "PUMP".

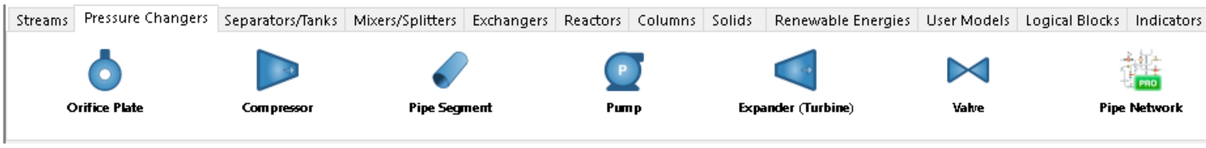
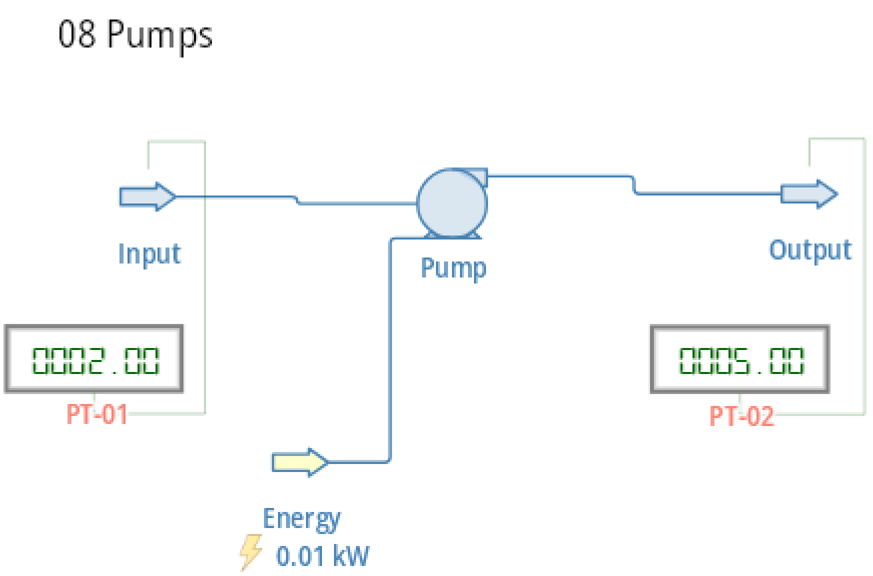


Figure 18 Pressure Changes Window

10. Click on "PUMP" block, the general information about the block is displayed on the right. Under "connections" tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose "Outlet Pressure". If all the connections are given correctly, all the blocks will turn blue.
11. Add pressure indicators on input and output stream and target the desired properties.
12. Run the simulation by clicking on "Solve flow sheet" icon / button on the top corner of the



screen.

Figure 19 Pump

13. To analyze/display the results, select on "Master property table" icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click "OK". The property table will be opened showing all the results as shown in the figure below.

	Output	**Input**
Temperature (K)	323.174	323.15
Pressure (Pa)	500000	200000
Mass Flow (kg/s)	0.0138889	0.0138889
Molar Flow (mol/s)	0.770951	0.770951
Volumetric Flow (m3/s)	1.40651E-05	1.40657E-05
Density (Mixture) (kg/m3)	987.473	987.432
Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	-2336.27	-2336.67
Specific Entropy (Mixture) (kJ/[kg.K])	-7.9599	-7.53883
Molar Enthalpy (Mixture) (kJ/kmol)	-42088.5	-42095.8
Molar Entropy (Mixture) (kJ/[kmol.K])	-143.4	-135.814
Thermal Conductivity (Mixture) (W/[m.K])	0.642833	0.642807

9 DETERMINATION OF POWER FOR A COMPRESSOR

Objective

Develop a simple process flow sheet to determine the power required to pressurize steam for a compressor.

Data

- Fluid: Water (Steam)
- Inlet mass flow rate: 50 kg/h
- Inlet Pressure: 2 bar
- Outlet Pressure: 5 bar
- Temperature: 125 °C

DWSIM Blocks Used

- Compressor
- Material stream
- Energy Stream
- Indicators

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on “New steady state Simulation” as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component “Water”
3. Specify the thermodynamic package as Raoult’s law.
4. Select the system of units for the simulation and click “Next”.
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as “Flow-In”. This serves as input stream.
6. Double click the “Flow-In” stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as “Flow-Out”. This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.
9. Below the Unit Operation tab on left, locate theCompressor block. Drag and drop into the flow sheet. Rename it as “COMPRESSOR”.

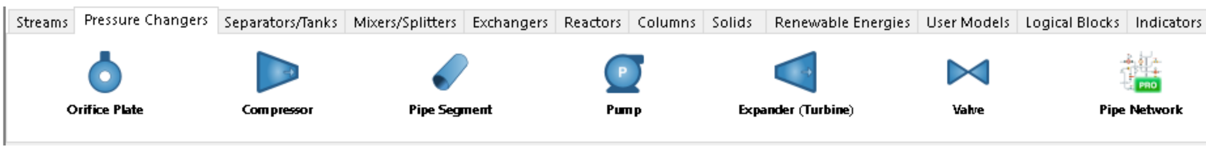
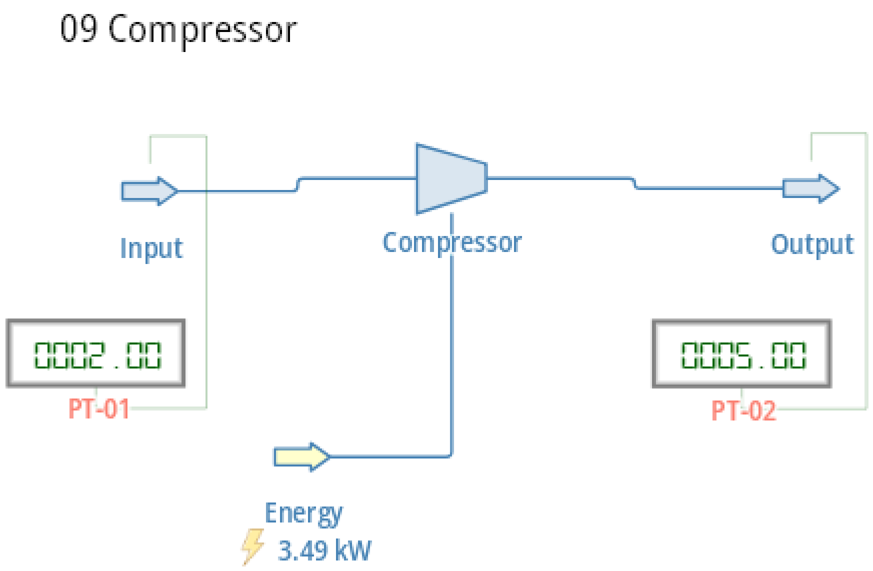


Figure 20 Pressure Changes window

10. Click on “COMPRESSOR” block, the general information about the block is displayed on the right. Under “connections” tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose “Outlet Pressure”. If all the connections are given correctly, all the blocks will turn blue.
11. Add pressure indicators on input and output stream and target the desired properties.
12. Run the simulation by clicking on “Solve flow sheet” icon / button on the top corner of the



screen.

Figure 21 Compressor

13. To analyze/display the results, select on “Master property table” icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click “OK”. The property table will be opened showing all the results as shown in the figure below.

	Input	**Output**
Temperature (K)	398.15	528.238
Pressure (Pa)	200000	500000
Mass Flow (kg/s)	0.0138889	0.0138889
Molar Flow (mol/s)	0.770951	0.770951
Volumetric Flow (m3/s)	0.0127601	0.00677168
Density (Mixture) (kg/m3)	1.08846	2.05103
Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	188.254	439.466
Specific Entropy (Mixture) (kJ/[kg.K])	0.230405	0.353053
Molar Enthalpy (Mixture) (kJ/kmol)	3391.45	7917.1
Molar Entropy (Mixture) (kJ/[kmol.K])	4.1508	6.36036
Thermal Conductivity (Mixture) (W/[m.K])	0.0265363	0.0391619

10 POWER GENERATED BY A TURBINE

Objective

Develop a simple process flow sheet to determine the power generated by a turbine.

Data

- Fluid: Water (Steam)
- Inlet mass flow rate: 50 kg/h
- Inlet Pressure: 5 bar
- Outlet Pressure: 2 bar
- Temperature: 125 °C

DWSIM Blocks Used

- Turbine
- Material stream
- Energy Stream
- Indicators

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on “New steady state Simulation” as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component “Water”
3. Specify the thermodynamic package as Raoult’s law.
4. Select the system of units for the simulation and click “Next”.
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as “Flow-In”. This serves as input stream.
6. Double click the “Flow-In” stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as “Flow-Out”. This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.
9. Below the Unit Operation tab on left, locate theTurbine block. Drag and drop into the flow sheet. Rename it as “TURBINE”.

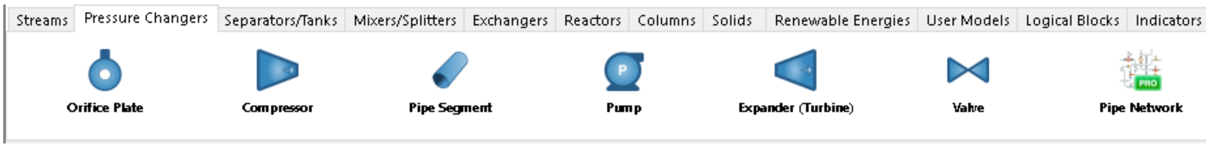
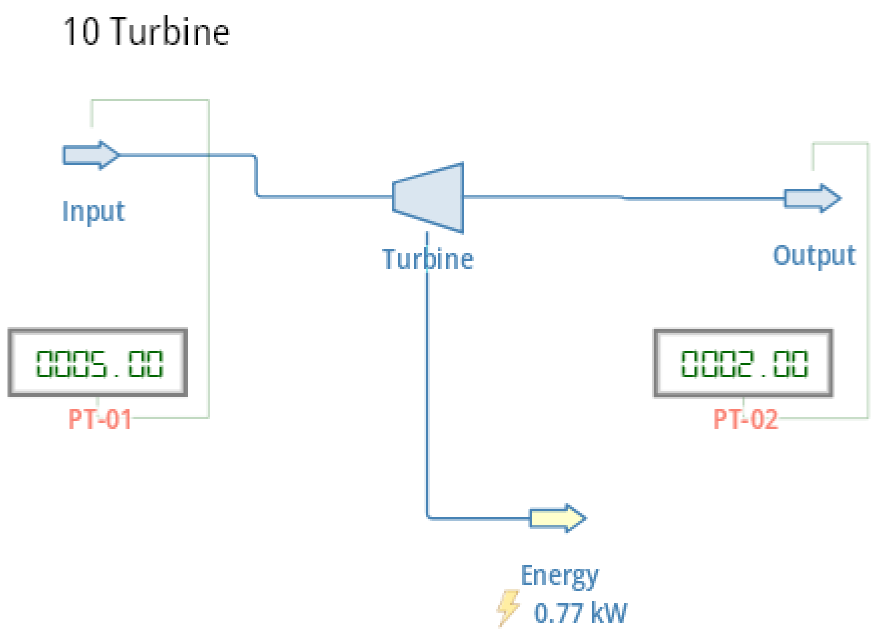


Figure 22 Pressure Changes Window

10. Click on "TURBINE" block, the general information about the block is displayed on the right. Under "connections" tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose "Outlet Pressure". If all the connections are given correctly, all the blocks will turn blue.
11. Add pressure indicators on input and output stream and target the desired properties.
12. Run the simulation by clicking on "Solve flow sheet" icon / button on the top corner of the



screen.

Figure 23 Turbine

13. To analyze/display the results, select on "Master property table" icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click "OK". The property table will be opened showing all the results as shown in the figure below.

	Output	**Input**
Temperature (K)	386.222	398.15
Pressure (Pa)	200000	500000
Mass Flow (kg/s)	0.0138889	0.0138889
Molar Flow (mol/s)	0.770951	0.770951
Volumetric Flow (m3/s)	1.46424E-05	1.47898E-05
Density (Mixture) (kg/m3)	948.538	939.084
Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	-2061.48	-2005.9
Specific Entropy (Mixture) (kJ/[kg.K])	-5.59367	-5.70333
Molar Enthalpy (Mixture) (kJ/kmol)	-37138.1	-36136.8
Molar Entropy (Mixture) (kJ/[kmol.K])	-100.771	-102.747
Thermal Conductivity (Mixture) (W/[m.K])	0.685443	0.687684

11 PRESSURE DROP ACROSS A VALVE

Objective

Develop a simple process flow sheet to determine the pressure drop across a valve.

Data

- Fluid: Water
- Inlet mass flow rate: 50 kg/h
- Inlet Pressure: 5 bar
- Outlet Pressure: 2 bar
- Temperature: 50 °C

DWSIM Blocks Used

- Valve
- Material stream
- Indicators

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on "New steady state Simulation" as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component "Water"
3. Specify the thermodynamic package as Raoult's law.
4. Select the system of units for the simulation and click "Next".
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as "Flow-In". This serves as input stream.
6. Double click the "Flow-In" stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as "Flow-Out". This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.
9. Below the Unit Operation tab on left, locate theValve block. Drag and drop into the flow sheet. Rename it as "VALVE".

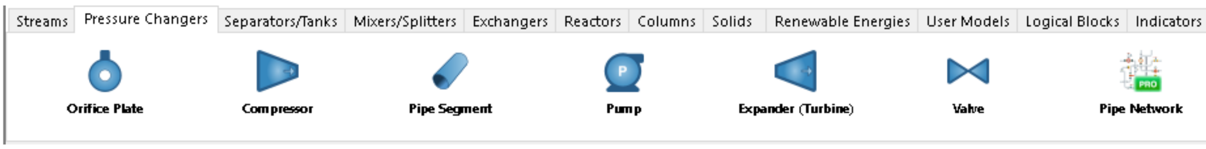


Figure 24 Pressure Changes Window

10. Click on “VALVE” block, the general information about the block is displayed on the right. Under “connections” tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose “Outlet Pressure”. If all the connections are given correctly, all the blocks will turn blue.
11. Add pressure indicators on input and output stream and target the desired properties.
12. Run the simulation by clicking on “Solve flow sheet” icon / button on the top corner of the screen.

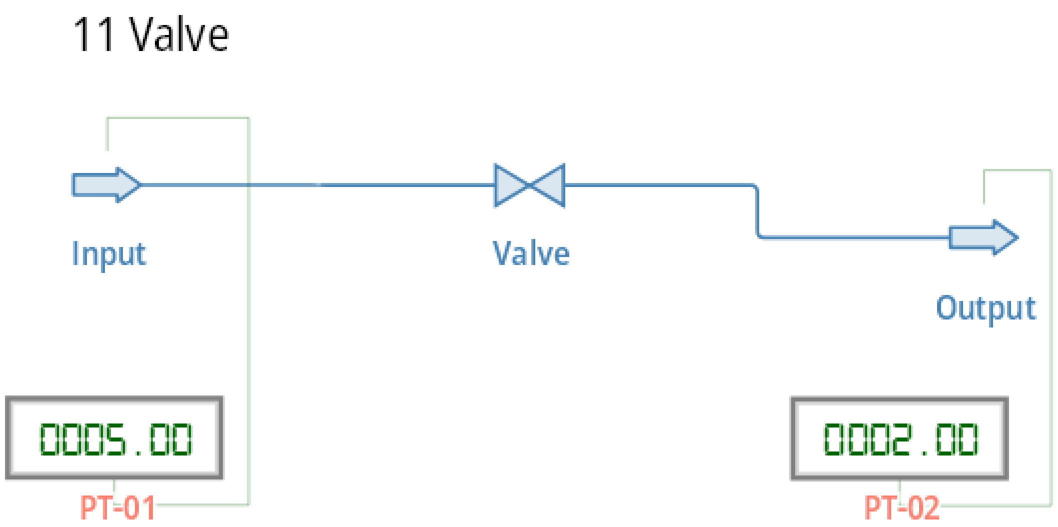


Figure 25 Valve

13. To analyze/display the results, select on “Master property table” icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click “OK”. The property table will be opened showing all the results as shown in the figure below.

	Output	**Input**
Temperature (K)	298.223	298.15
Pressure (Pa)	200000	500000
Mass Flow (kg/s)	0.0138889	0.0138889
Molar Flow (mol/s)	0.770951	0.770951
Volumetric Flow (m3/s)	1.39402E-05	1.39392E-05
Density (Mixture) (kg/m3)	996.323	996.389
Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	-2440.55	-2440.55
Specific Entropy (Mixture) (kJ/[kg.K])	-8.49746	-8.92232
Molar Enthalpy (Mixture) (kJ/kmol)	-43967.2	-43967.2
Molar Entropy (Mixture) (kJ/[kmol.K])	-153.084	-160.738
Thermal Conductivity (Mixture) (W/[m.K])	0.610357	0.610248

12 RESIDENCE TIME CALCULATION FOR A TANK

Objective

Develop a simple process flow sheet to determine residence time for a given volume of tank.

Data

- Fluid: Water
- Inlet mass flow rate: 500 kg/h
- Inlet Pressure: 1 bar
- Outlet Pressure: 1 bar
- Temperature: 50 °C
- Tank Volume: 5 m^3

DWSIM Blocks Used

- Tank
- Material stream
- Indicators

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on "New steady state Simulation" as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component "Water"
3. Specify the thermodynamic package as Raoult's law.
4. Select the system of units for the simulation and click "Next".
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as "Flow-In". This serves as input stream.
6. Double click the "Flow-In" stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as "Flow-Out". This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.

9. Below the Unit Operation tab on left, locate the Tank block. Drag and drop into the flow sheet. Rename it as "TANK".

Figure 26 Separator and tank window

10. Click on "TANK" block, the general information about the block is displayed on the right. Under "connections" tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose "Tank Volume". If all the connections are given correctly, all the blocks will turn blue.
11. Add flow indicators on input and output stream and target the desired properties and add time indicator on tank.
12. Run the simulation by clicking on "Solve flow sheet" icon / button on the top corner of the screen.

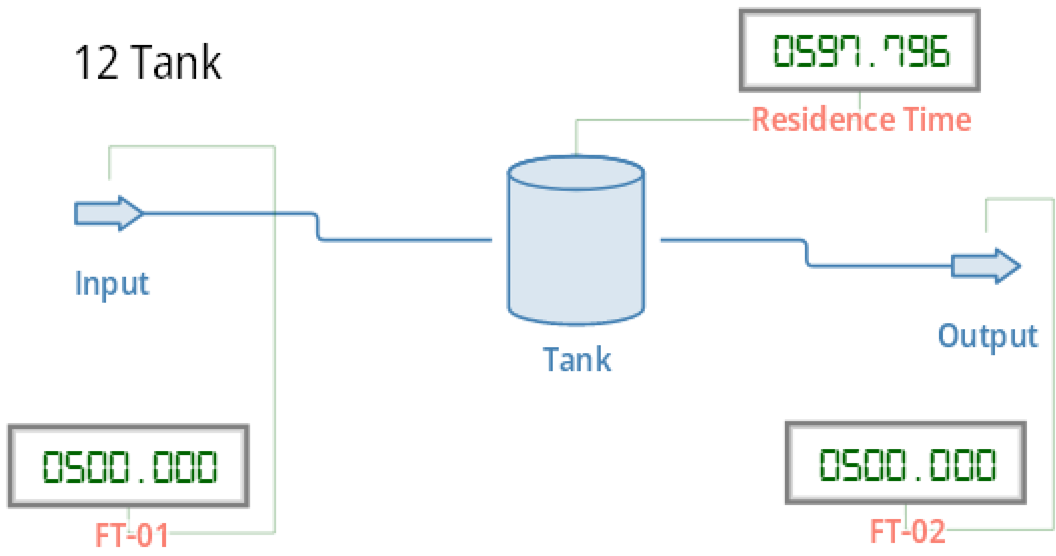


Figure 27 Tank

13. To analyze/display the results, select on "Master property table" icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click "OK". The property table will be opened showing all the results as shown in the figure below.

	Input	Output
Temperature (K)	298.15	298.15

Pressure (Pa)	101325	101325
Mass Flow (kg/s)	0.138889	0.138889
Molar Flow (mol/s)	7.70951	7.70951
Volumetric Flow (m3/s)	0.000139401	0.000139401
Density (Mixture) (kg/m3)	996.327	996.327
Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	-2440.95	-2440.95
Specific Entropy (Mixture) (kJ/[kg.K])	-8.18698	-8.18698
Molar Enthalpy (Mixture) (kJ/kmol)	-43974.4	-43974.4
Molar Entropy (Mixture) (kJ/[kmol.K])	-147.491	-147.491
Thermal Conductivity (Mixture) (W/[m.K])	0.610248	0.610248

13 Flow Split with Splitter

Objective

Develop a simple process flow sheet to split a given mass flow into equal proportions.

Data

- Fluid: Water
- Inlet mass flow rate: 90 kg/h
- Inlet Pressure: 1 bar
- Temperature: 50 °C

DWSIM Blocks Used

- Splitter
- Material stream
- Indicators

Procedure

1. Start a new DWSIM Simulation (DWSIM VER 8.0 - Classic UI). Click on "New steady state Simulation" as a template for new simulation
2. The simulation configuration window will be opened. It shows a specification page. Add the component "Water"
3. Specify the thermodynamic package as Raoult's law.
4. Select the system of units for the simulation and click "Next".
5. The flow sheeting section of simulation window will be opened. Drag and drop the Material stream available at the right, in the object palette. Rename the stream as "Flow-In". This serves as input stream.
6. Double click the "Flow-In" stream. The general information about the stream will be displayed on the right side of screen. Specify the feed compositions, flow rate, temperature, and pressure for the inlet streams once composition and flow rate are specified for the inlet streams, the color of stream turns blue.
7. Add one more Material stream i.e., Drag and drop it into the flow sheet. Rename the stream as "Flow-Out". This serves as the output stream.
8. Add an energy stream from the object palette to the flow sheeting section.
9. Below the Unit Operation tab on left, locate the Splitter block. Drag and drop into the flow sheet. Rename it as "SPLITTER".

Figure 28 Mixer and Splitters Window

10. Click on “SPLITTER” block, the general information about the block is displayed on the right. Under “connections” tab, for all streams click the dropdown button and select the necessary connections. Under calculation type choose “Tank Volume”. If all the connections are given correctly, all the blocks will turn blue.
11. Add flow indicators on input and output stream and target the desired properties.
12. Run the simulation by clicking on “Solve flow sheet” icon / button on the top corner of the screen.

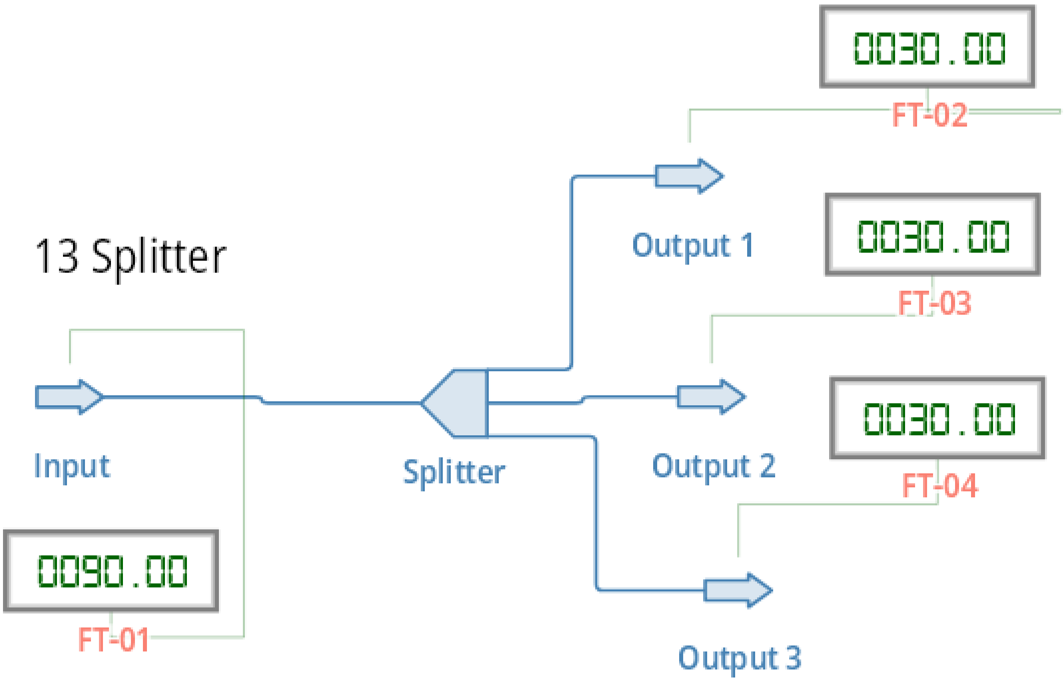


Figure 29 Splitter

13. To analyze/display the results, select on “Master property table” icon on the tool bar. A box will appear which is double clicked to modify it further. Select the streams which have to be shown in output and click “OK”. The property table will be opened showing all the results as shown in the figure below.

	Input	**Output 1**	**Output 2**	**Output 3**
Temperature (K)	298.15	298.15	298.15	298.15
Pressure (Pa)	500000	500000	500000	500000
Mass Flow (kg/s)	0.025	0.00833333	0.00833333	0.00833334
Molar Flow (mol/s)	1.38771	0.46257	0.46257	0.462571
Volumetric Flow (m3/s)	2.50906E-05	8.36353E-06	0.00215098	0.00215099
Density (Mixture) (kg/m3)	996.389	996.389	3.87419	3.87419
Molecular Weight (Mixture) (kg/kmol)	18.0153	18.0153	18.0153	18.0153
Specific Enthalpy (Mixture) (kJ/kg)	-2440.55	-2440.55	-151.963	-151.963

Specific Entropy (Mixture) (kJ/[kg.K])	-8.92232	-8.92232	-1.24636	-1.24636
Molar Enthalpy (Mixture) (kJ/kmol)	-43967.2	-43967.2	-2737.65	-2737.65
Molar Entropy (Mixture) (kJ/[kmol.K])	-160.738	-160.738	-22.4536	-22.4536
Thermal Conductivity (Mixture) (W/[m.K])	0.610248	0.610248	0.0550495	0.0550495