Tutorials

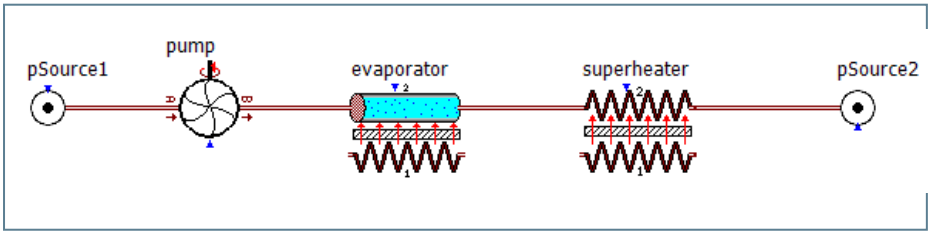
Tutorial 4 - Evaporator Super Heater

Objective

With the aim of this tutorial a model of a simple two-phase heat exchanger will be created. In a first step the model will be set up purely to represent the physical behavior of the system. In a second step, means to perform some tests are added.

The purpose of the circuit is to evaporate the water in a first step and super heat it subsequently in the next step. The modeled system may be part of a steam-power process.

- *Multi domain modeling*
- *Result analysis and balance tests*
- *Parameter studies*



It is assumed that you are familiar with the basic functionality of SimulationX. Therefore, please refer to "Tutorial 1: Getting Started" for a general introduction on how to select Elements from the libraries, how to connect them and enter parameters, how to run a simulation and how to open result windows.

Part 1: Open-Loop System

Create the SimulationX model of the Evaporator Super Heater according to

Figure 1.

Use the elements and the names (blue market in brackets) listed in Table 1. You can change the label text and label position of each object by double clicking on an element and selecting "General/Name …". You have to write the name without a blank (e.g. "superheater"). Some elements have to be rotated before connecting.

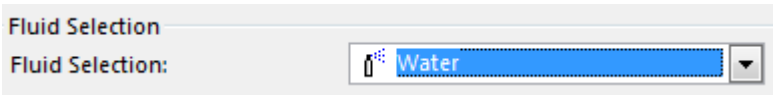
When connecting the elements with each other, you should remember, that in SimulationX you can only connect element ports of the same type.

Figure 1: Model structure of the Evaporator Super heater configuration

The path of a connection will be determined automatically, but changing the path is possible at any time. To do this, move the mouse over a connection, while pressing the Alt-key. The mouse pointer shows you in which direction you can move the selected connection line.

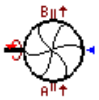
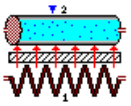
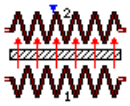
In order to cut an already existing connection, click on it and press "Del".

Once all elements have been connected properly choose one connection and right click on it. In the appearing context menu select properties and the property dialog will be shown. In the "Fluid" tab the fluid has to be changed from default "Air_IDG" to "Water".



This has to be done only once for an arbitrary connection since the fluid information is propagated automatically to all other connections within the circuit.

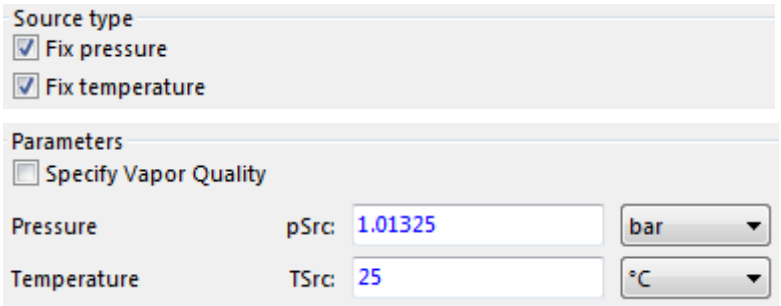
Table 1: Elements required for the circuit in Figure 1

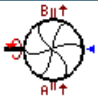
Number of elements	*Library name*	*Element name*	*Symbol*
2	*Thermal-Fluid*	*Pressure Supply (pSource1)*	
1	*Thermal-Fluid*	*Dynamic Transformer (pump)*	
1	*Thermal-Fluid*	*Evaporator (evaporator)*	
1	*Thermal-Fluid*	*Heat Exchanger (superheater)*	

Once you have succeeded in creating the model structure according to

Figure 1, you have to enter the parameters for the elements and activate the protocol attribute for the variables which you want to plot after the simulation. Since almost all elements have default parameters, you only have to enter the parameters which are different from the default values. Table 2 gives an overview about the parameter settings which you have to enter. Some of the parameter dialogs have more than one page.

Table 2: Parameter settings

Element	*Parameter Input*
pSource1 	• *Leave all default values as they are. Note that the check box for the vapor quality is not checked because the fluid will enter the evaporator purely liquid.* 
pump	• *Leave the default for the transformation type:* 



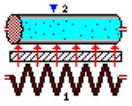
- *Change the flow configuration to mass flow.*

Flow Configuration	flow1:	Mass Flow

- *Set the mass flow to 0.6 kg/min. Don't forget to change the unit.*

Mass Flow from A to B	Qm0:	0.6	kg/min

evaporator



In the Tab Global Parameter leave the default Specification Mode and set:

- *the heat Power to 30kW.*

Specification Mode	mode:	Heat power specified
Heat Power Setting	Pw0:	30 kW

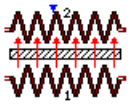
In SimulationX one activates the result attributes by changing the protocol icons.  *->* 

- *Activate the protocol attributes for the Heat Transfer from side 1 to 2:*

Heat transferred from sid... phi:	W

In the other tabs leave the default values unchanged.

superheater



In the Tab Global Parameter leave the default Specification Mode and set:

- *the heat Power to 5kW.*

Specification Mode	mode:	Heat power specified
Heat Power Setting	Pw0:	5 kW

- *Activate the protocol attributes for the Heat Transfer from side 1 to 2:*

Heat transferred from sid... phi:	W

In the other tabs leave the default values unchanged.

pSource2

Dialog page "Parameters" :

- *Leave the check box "Fix pressure" as it is but uncheck "Fix temperature". Note that the check box for the vapor quality is also not checked because the fluid will leave the system here no matter in which phase. Set the pressure to 8 bar.*

☑ Fix pressure			
☐ Fix temperature			
Pressure	pSrc:	8	bar

connection2

Dialog page "Parameters", setting initial conditions:

- *Leave the default value of the temperature but set po to pSource2.pSrc.*

Initial Pressure	p0:	pSource2.pSrc	Pa
Initial Temperature	T0:	25	°C

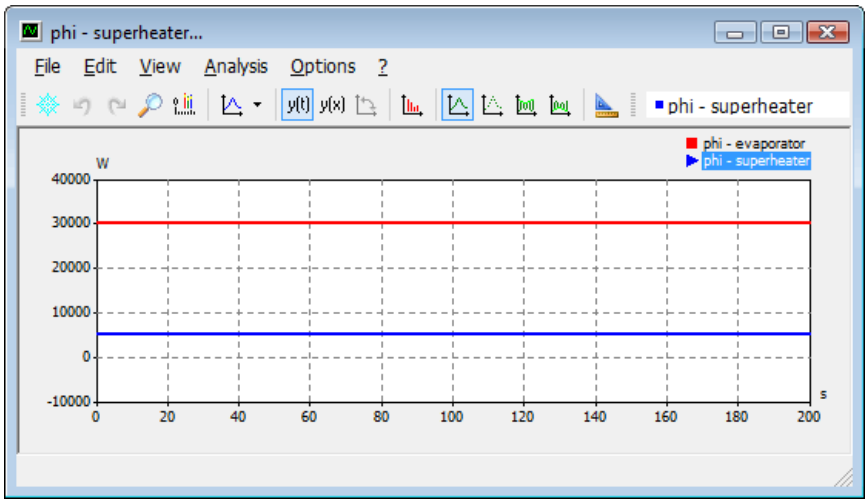
connection3

Dialog page "Parameters", setting initial conditions:

- *Leave the initial value of the temperature but set the pressure po to the source pressure of element pSource2.*

	Initial Pressure p0: pSource2.pSrc Pa Initial Temperature T0: 25 °C
Simulation properties	*Set the Simulation Stop time to 200 s.* Stop Time tStop: 200 s

Once you have entered the parameters of the model, you can run the simulation and observe the results. Figure 2 shows some of the simulation results. A result window will be opened if one drags the icon of an activated protocol attribute into the model view. In order to show two results curves in just one result window the icon of a second protocol attribute has to be dragged into the existing result window.



The values of the transferred heat fluxes within the evaporator and superheater respectively are illustrated.

Figure 2: Transferred heat flux in the evaporator and superheater

- *You can play with the simulation model, in order to get some feeling how the model will react on parameter changes. (Don't forget to reset the model before you make modifications!)*

Part 2: Performing some tests

Now it will be described how some balance tests can be carried out in order to check the results for correctness. It will be verified whether the increase of the enthalpies will be correspond with the transferred heat fluxes or not. To this end some state sensors and two function blocks will be added.

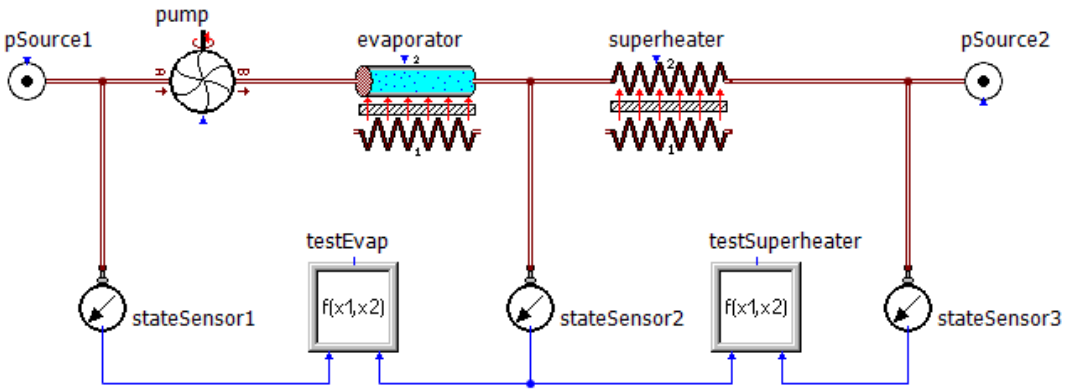


Figure 3: Modified model structure

To add these elements to the model, carry out the following steps:

- *Reset the simulation model. ( )*

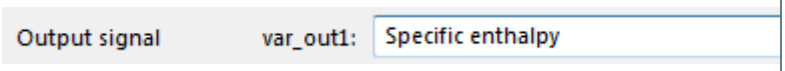
- *Add three state sensors and connect them with the existing model according to Figure 3. Add also two function elements with two inputs from the Signal Block library. Rename those elements as suggested.*

Table 3: Accumulator parameters

<table>
<tr>
<td>State Sensor
(stateSensor1; stateSensor2;
stateSensor3)

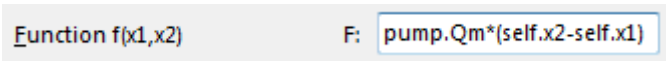
  </td>
<td>Dialog page "Parameters" :

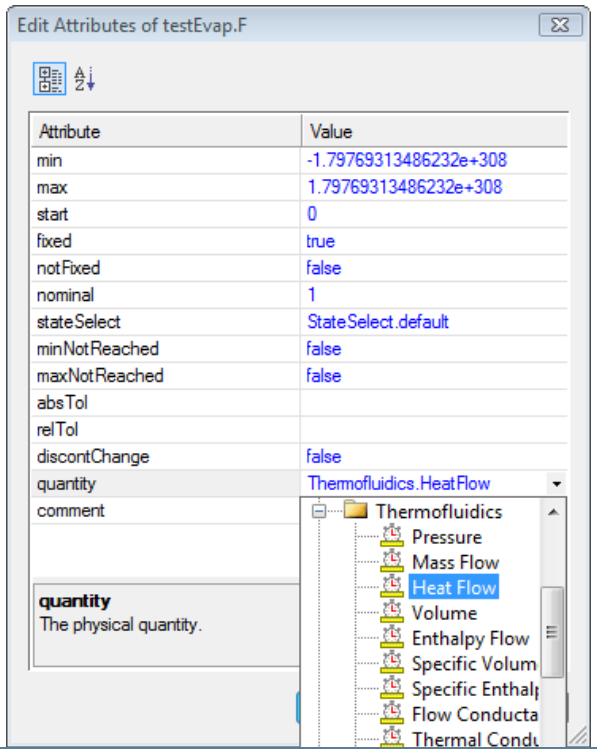
• Set the output signal to "Specific enthalpy". Note, you can select all three stateSensors by clicking on the elements while the control-key is pressed. Then a double click on one element will open the property dialog and the appropriate output signal needs to be set only once.

  </td>
</tr>
<tr>
<td>Function

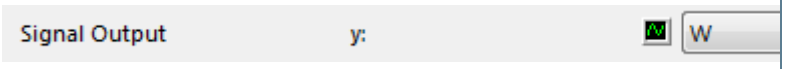
(testEvap; testSuperheater)

  </td>
<td>• In the dialog page "Parameters" type in the following which corresponds to the increase of the enthalpy flux. x2 and x1 hold the values of the enthalpies at the input and the output side of the Evaporator and Superheater respectively, see the state sensors. The mass flow through the heat exchangers is equal to that of the pump when the simulation reaches a steady state.

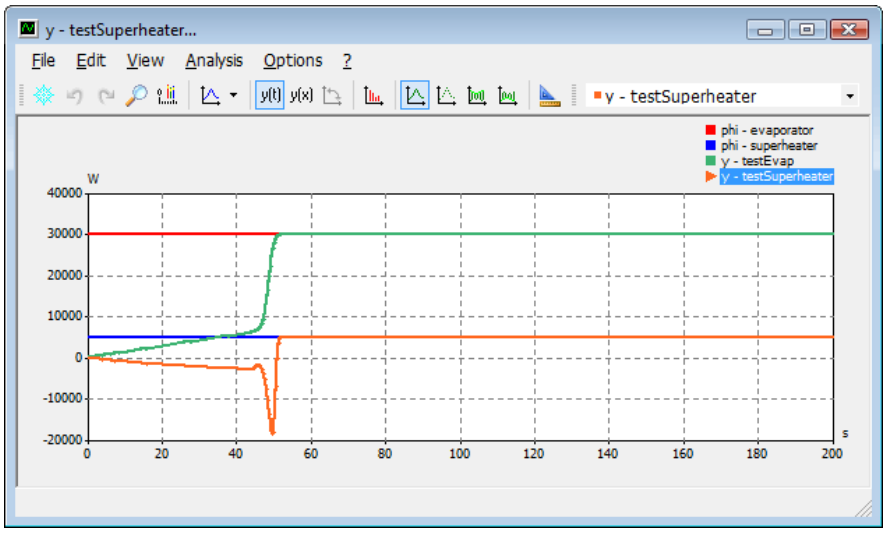
 

Select the attributes by using the button  and change the quantity from the default to Thermofluidics - heat flow for the parameter and the result variable.</td>
</tr>
<tr>
<td></td>
<td>• Activate the protocol attributes for the Signal Output y:

Select as unit 'W'.</td>
</tr>
</table>



Drag the result icons of the function elements into the already existing result window, Figure 2.

Run the simulation and compare the values of the transferred heat flux with the corresponding enthalpy flux increase.

Figure 4: Comparison of heat flux and enthalpy flux increase

In the figure above it is well discernable that after reaching the steady state the energy balance is fulfilled.

Finally, let us resume a few points concerning the benefits of this tutorial

- *You have learned how to construct models of Thermofluidic systems in SimulationX. Most of the elements have default parameters and scalable options.*
- *Signal blocks in SimulationX (e.g. signal sources) can be adapted to your purpose in terms of physical quantities, units and names of parameters and result quantities. Above that they can be used to perform balance and other checks.*
- *SimulationX is a tool for intuitive system simulation, since models with mixed physical domains (e.g. mechanics, hydraulics, thermofluidics and controls) can be created very quickly.*
- *The object-oriented approach of SimulationX allows you to modify a given model structure very easily and adapt it to new tasks and specifications.*
- *The SimulationX model clearly resembles the circuit structure of the thermofluidic system. This means, that you can concentrate on your engineering task instead of thinking about the mathematical background of your system. Moreover, other engineers can easily re-use and understand your own models.*