A CAPE-OPEN compliant simulation module

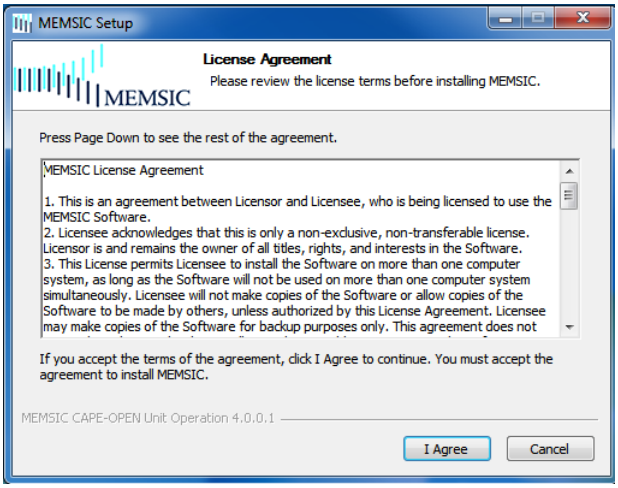
Notice

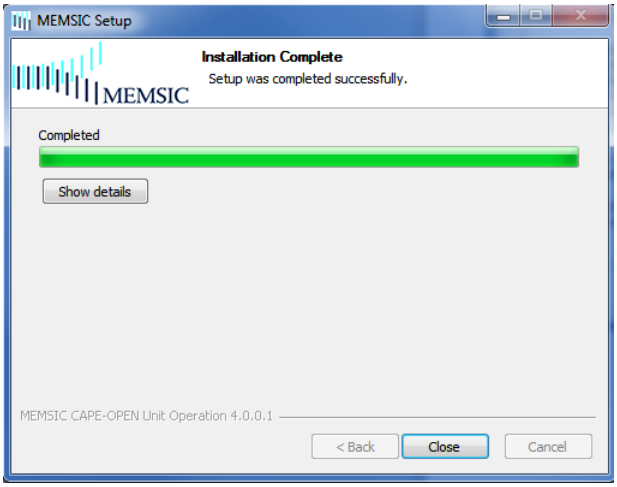
MEMSIC is a useful CAPE-OPEN compliant simulation software to simulate gas separation processes through a membrane module. This software has been developed at the Separation Processes Group (Laboratoire Réactions et Génie des Procédés, UMR CNRS 7274 – Nancy, France). Most process simulation software tools (PSE: Process Simulation Environment as ASPEN®, HYSYS®, PRO/II ®, PROSIM®, etc.) implement CAPE-OPEN interfaces that enables the end-user to plug CAPE-OPEN compliant tools and to export CAPE-OPEN compliant components.

Four different types of hydrodynamic conditions are taken into account in MEMSIC: (1) cross plug flow, (2) perfect mixing, (3) Co-Current plug flow and (4) Counter-Current plug flow. Additionally, five different concepts and/or theoretical model are proposed to describe the transport mechanism of molecular species through a membrane: (1) Constant Permeability, (2) Dual Mode, (3) Henry, (4) ENSIC and (5) Flory-Huggins.

Installation of the CAPE-OPEN module

In order to install the module, we have developed a wizard. By clicking on the setup assistant and after accepted the license agreement, the installation goes through a series of dialog boxes. **Don't forget to plug the dongle into your computer before install the program.**

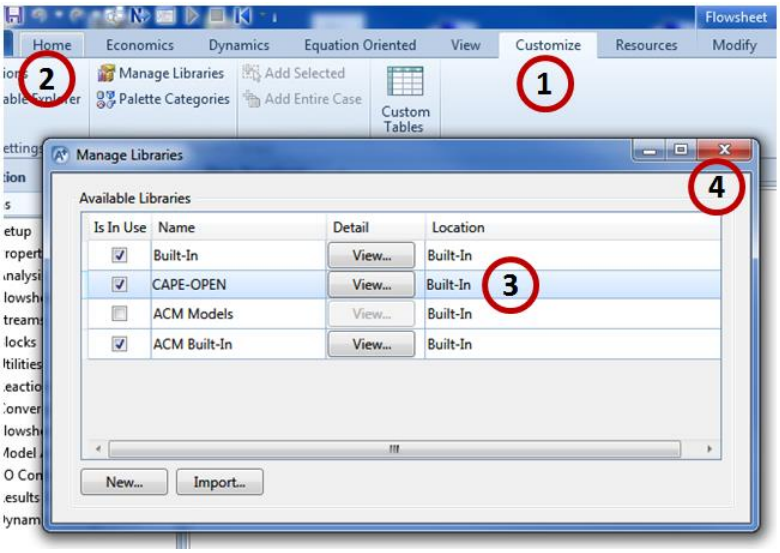


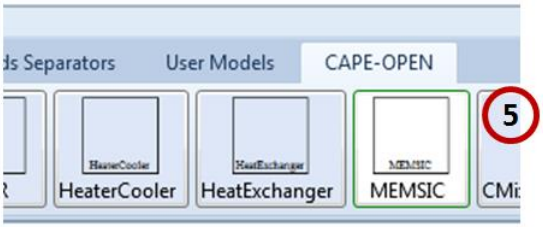


Implementation of the module in AspenPlus®

Four steps are needed:

1. Click “Customize” on the menu bar
2. Click “Manage Librairies” on the toolbars
3. Activate the “CAPE-OPEN” module
4. Close the window





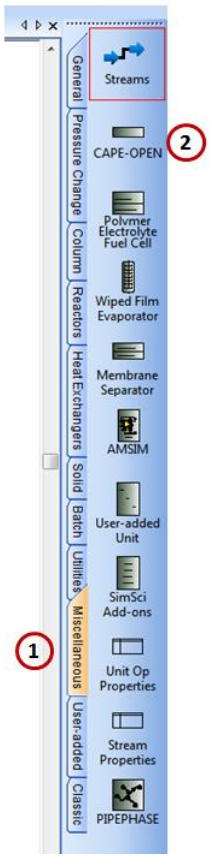
Then, after these steps, a new model library appears at the bottom: “CAPE-OPEN”. Use this new model library to select the unit operation model MEMSIC and place it on the flowsheet.

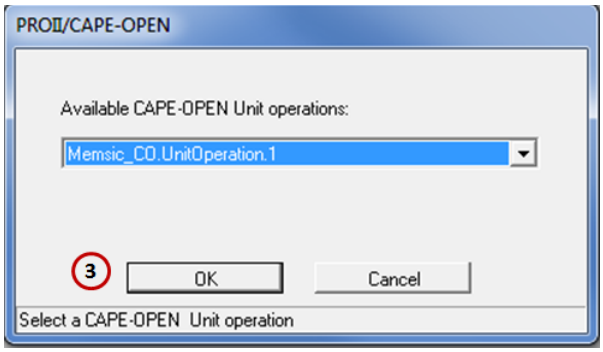
Implementation of the module in PRO/II®

Three steps are needed:

1. Click “Miscellanous” on the menu bar
2. Clik on the “CAPE-OPEN” icon
3. Select the MEMSIC module

Then, drag and drop the icon on the flowsheet.



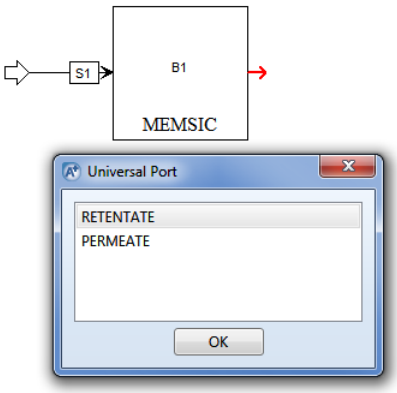


Utilization of MEMSIC

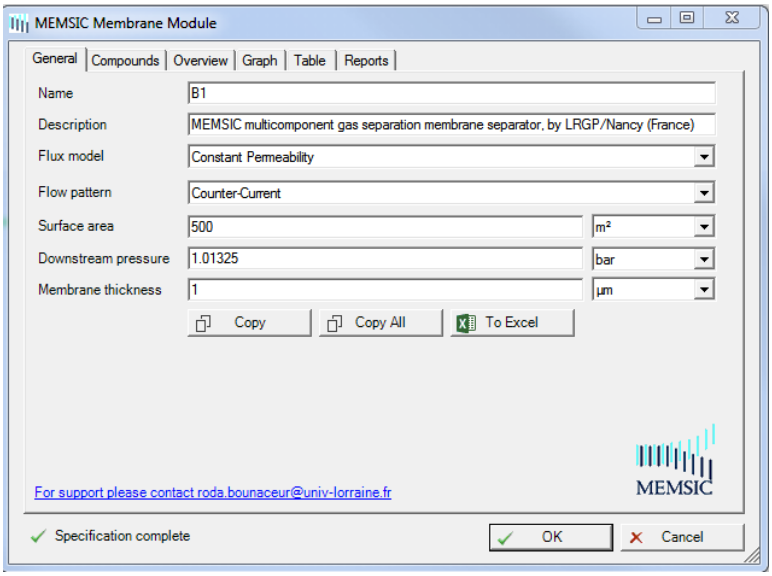
Whatever the PSE, you have to follow these required steps:

1. Select the components to be used in your model
2. Set the calculation methods for physical and thermodynamical properties
3. Create your flowsheet by adding object: unit operation, streamline, compressor, etc.

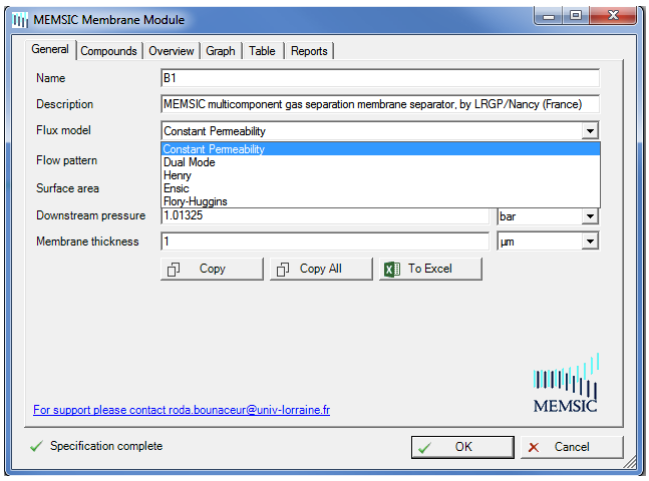
When you connect a material stream with the module MESIC, in new windows appears in order to select if the stream has to be considered as the retentate side or the permeate side

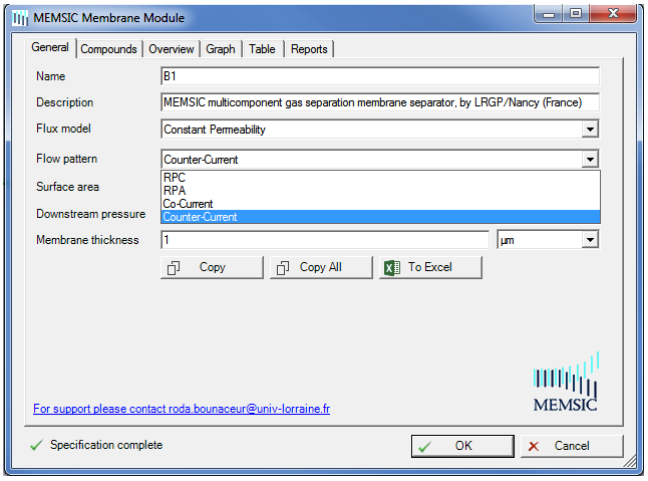


4. Specify Material Streams to fully define both the stream's composition and its thermodynamic state: pressure, mole fraction, flow rate, etc.
5. When you double click on the MEMSIC module, a new windows opens with 6 different items: "General", "Compounds, Overview, Graph, Table, Reports

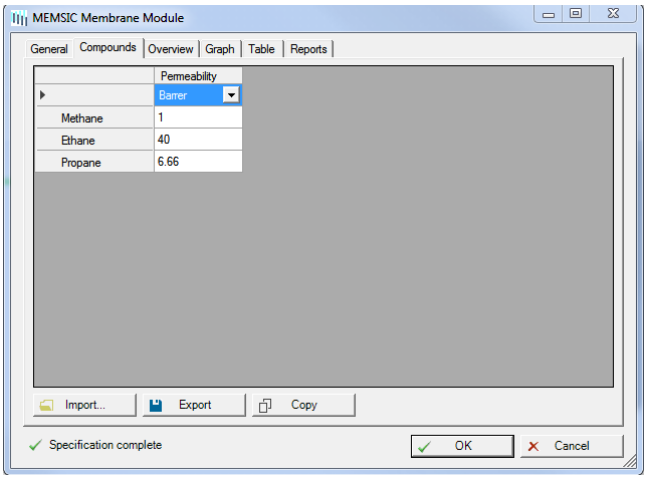


6. "General": enter the operating parameters, the surface area, the thickness, the flux model and the flow pattern

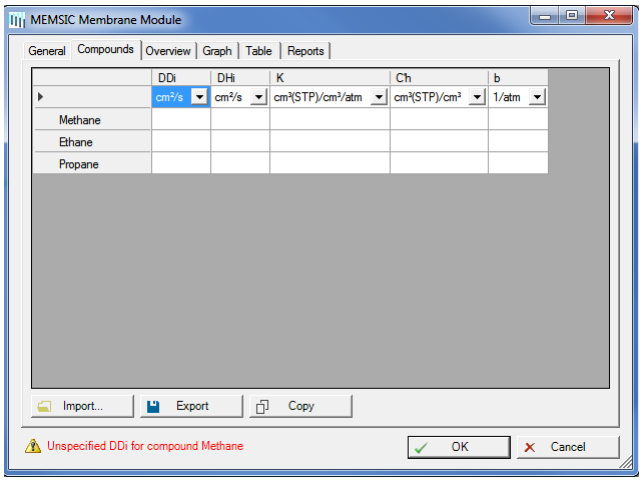




7. "Compound": depending on the flux model, enter the different necessary parameters

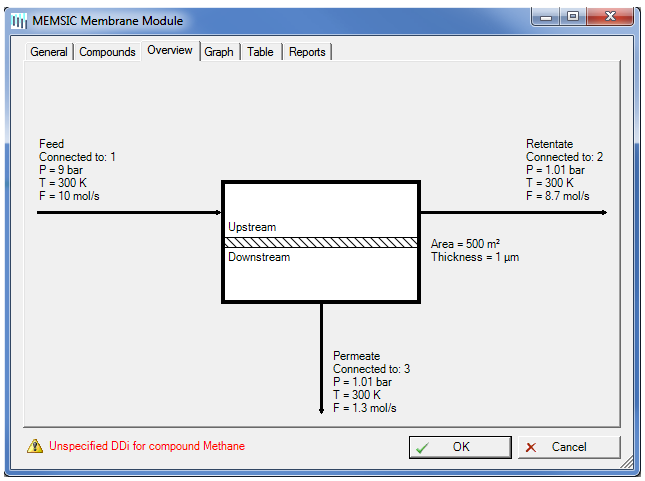


(Constant permeability)

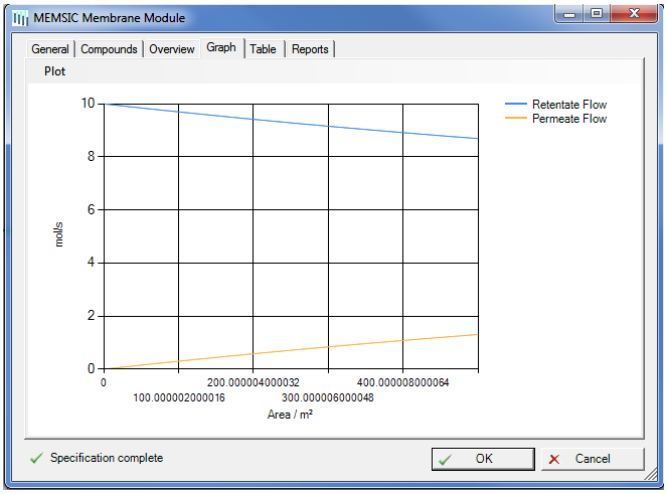


(Dual Mode)

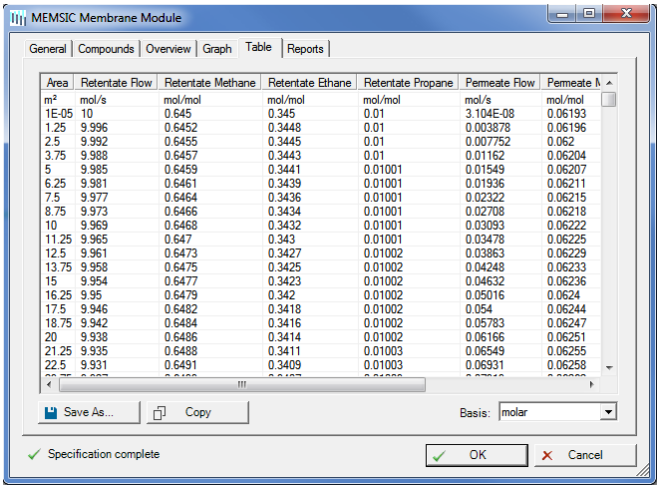
8. "Overview"/"Graph"/"Table"/"Reports": when the calculation is finished, this tab give an overview of the results



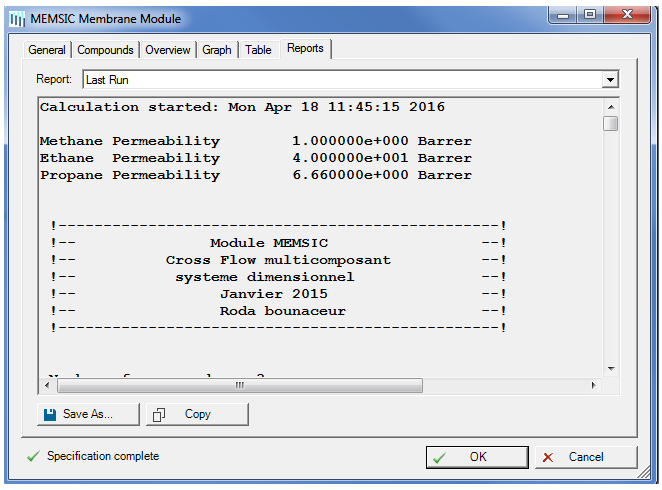
"Overview"



"Graph"



"Table"



"Reports"

9. By clicking on the "To Excel" button, you store all the results in an excel file