How to use MEMSIC®

MEMSIC®, a CAPE-OPEN compliant simulation module

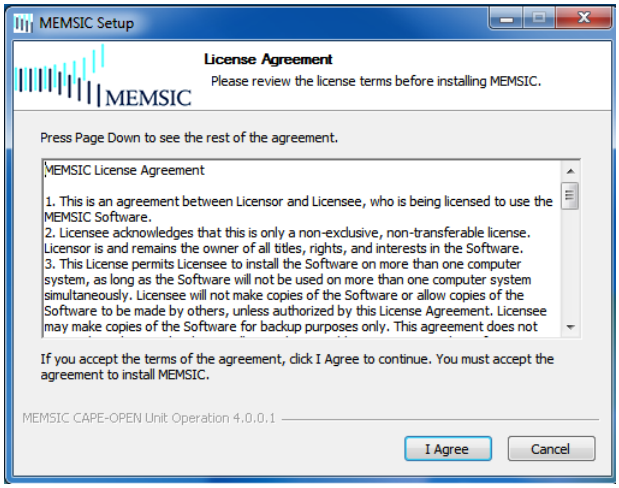
MEMSIC® is a useful CAPE-OPEN compliant simulation software to simulate gas separation processes through a membrane module. This software has been developed at the Separation Processes Group (Laboratoire Réactions et Génie des Procédés, UMR CNRS 7274 – Nancy, France). Most process simulation software 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.

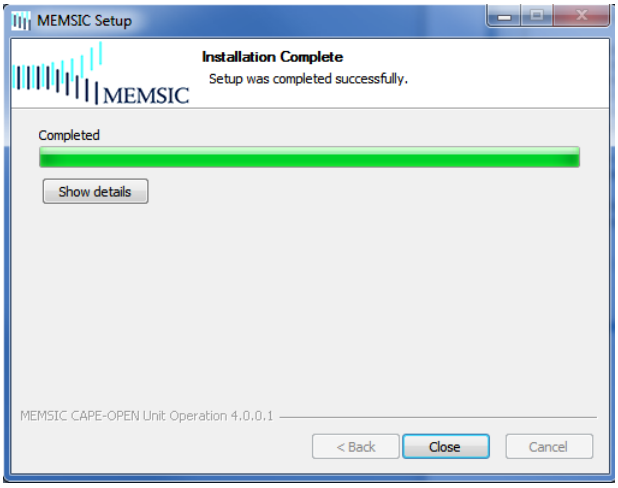
Four different types of hydrodynamic conditions are taken into account in MEMSIC®: (1) cross plug flow, (2) perfect mixing, (3) Co-Current plug flow and (4) Counter-Current plug flow. Additionally, five different concepts and/or theoretical model are proposed to describe the transport mechanism of molecular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

This notice explains how to install the software on your computer and activate the MEMSIC® unit operation on several Process Simulation Software, and how to use MEMSIC®.

Installation of the CAPE-OPEN module

To install the MEMSIC® module, we have developed a setup assistant. By clicking on the setup assistant, and after having accepted the license agreement, the installation goes through a series of dialog boxes. **Do not forget to plug the dongle into your computer before installing the program.**

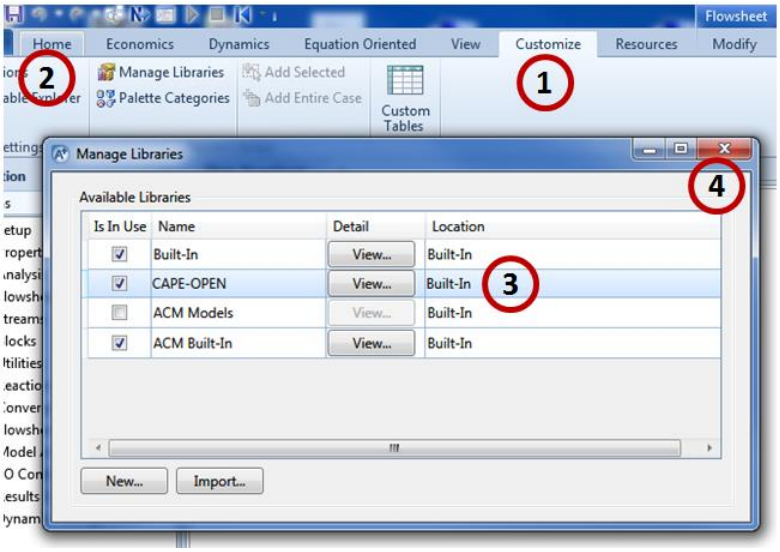


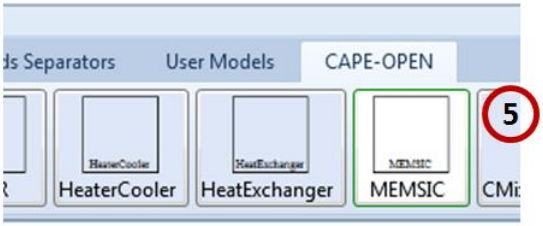


How to use MEMSIC®

Implementation of the module in AspenPlus®

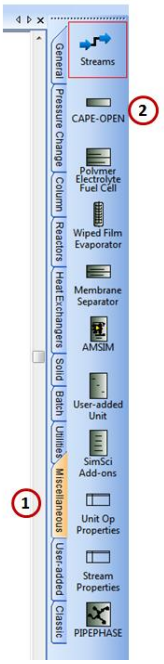
1. Click on “Customize” on the menu bar.
2. Click on “Manage Librairies” on the toolbar.
3. Activate the “CAPE-OPEN” module.
4. Close the window. A new model library appears at the bottom toolbar, named “CAPE-OPEN”. Click on this new tab, and select “MEMSIC” module. You can then place a new MEMSIC® unit operation on the flowsheet.

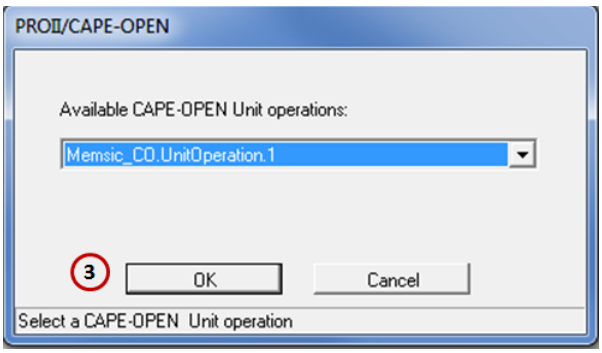




Implementation of the module in PRO/II®

1. Click “Miscellaneous” on the menu bar.
2. Click on the “CAPE-OPEN” icon.
3. Select the “MEMSIC” module. Then, drag and drop the icon on the flowsheet to create a MEMSIC® unit operation on your flowsheet.





How to use MEMSIC®

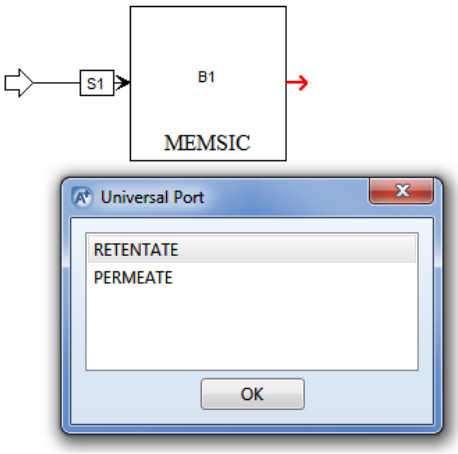
How to use MEMSIC

Whatever the PSE you choose, you have to follow these required steps to use the MEMSIC® module:

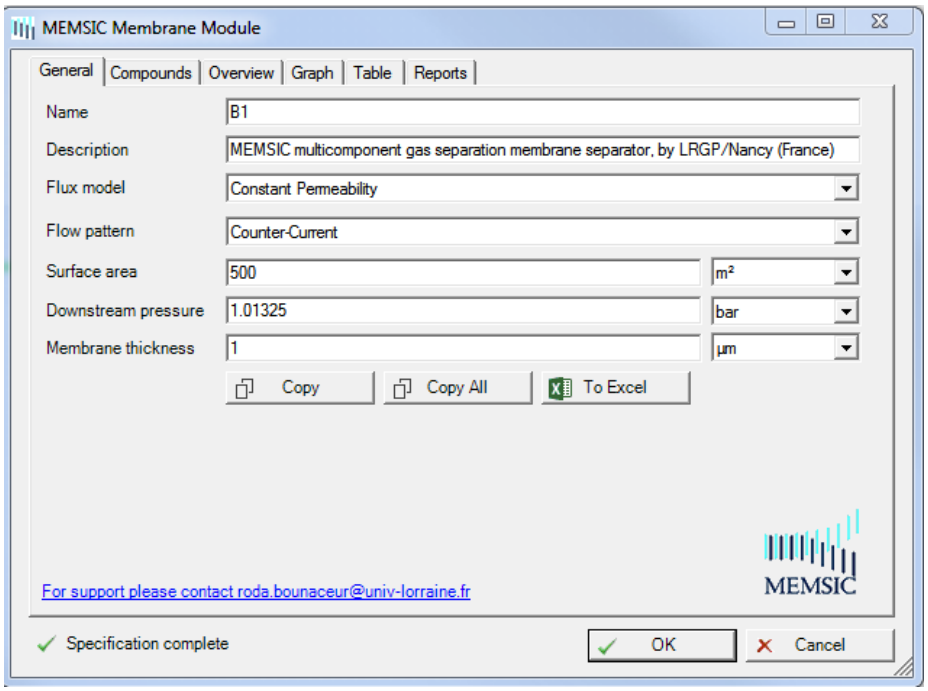
1. Select the components to be used in your model
2. Set the calculation methods for physical and thermodynamics properties
3. Create your flowsheet by adding object: unit operation, streamline, compressor, etc.
4. Define input and outputs streams of the MEMSIC® module.

Define the Material Streams by defining both the stream's composition and its thermodynamic state: pressure, mole fraction, flow rate, etc...

The MEMSIC® module must be connected to one inlet stream and two outlet streams. When you create an output Material Stream from the MEMSIC® unit operation, a message box appears and ask you to specify which outlet this stream is: retentate or permeate.

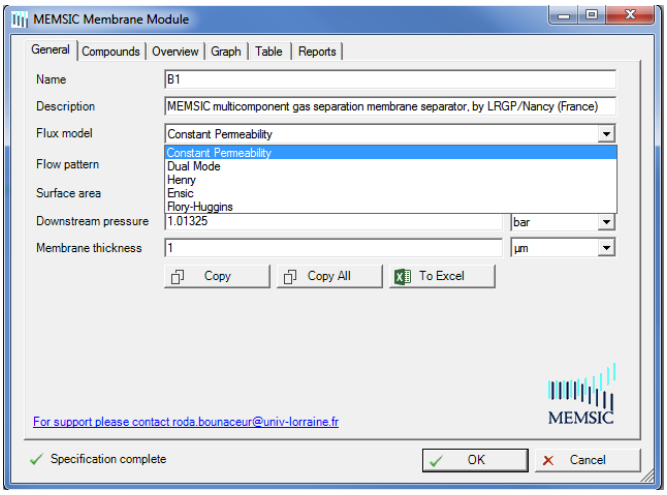


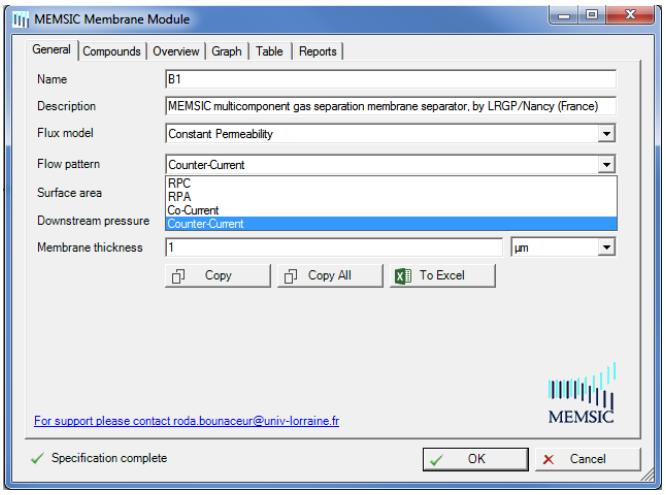
5. When double clinking on the MEMSIC module, a new window opens with 6 different tabs: "General", "Compounds", "Overview", "Graph", "Table", and "Reports".



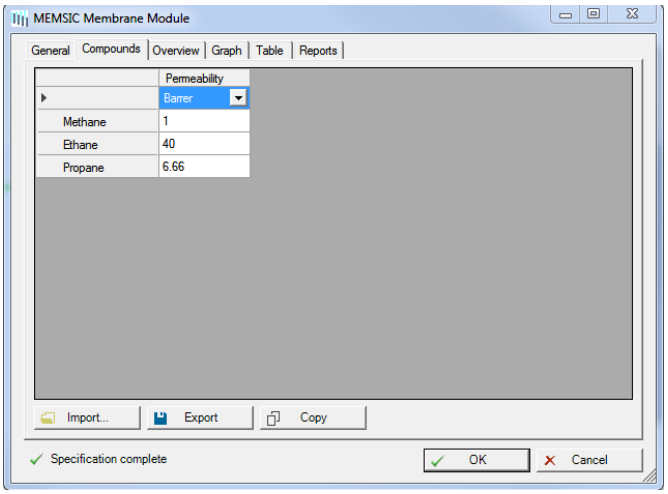
How to use MEMSIC®

6. Select the "General" tab, select the flux model and the flow pattern, and then define the operating parameters: surface area, membrane thickness, and downstream pressure.

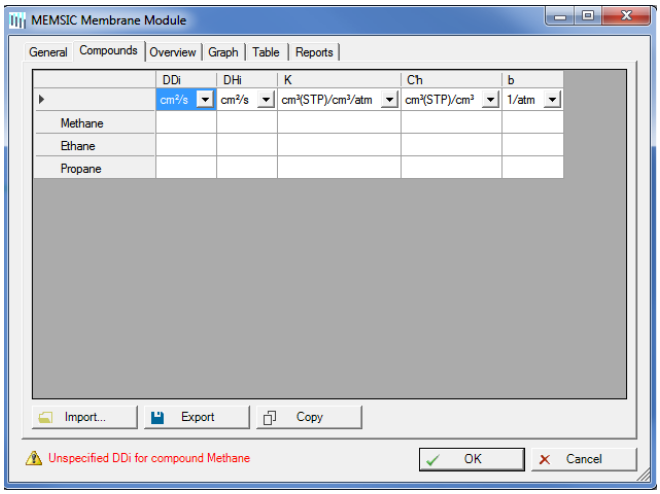




7. Select the "Compound" tab, enter the required parameters depending on the selected flux model.



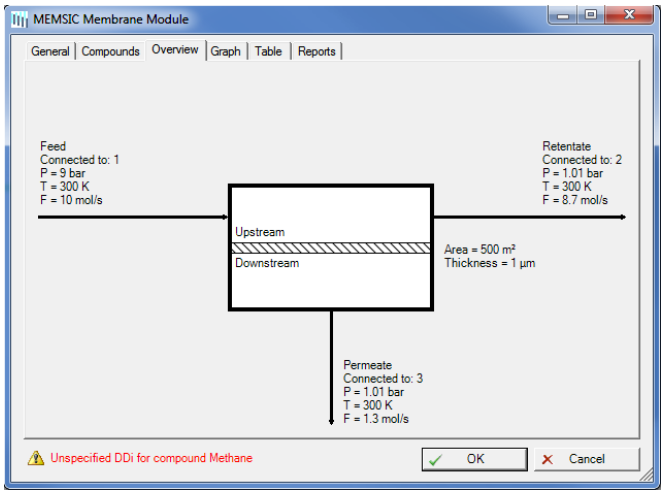
(Constant permeability)



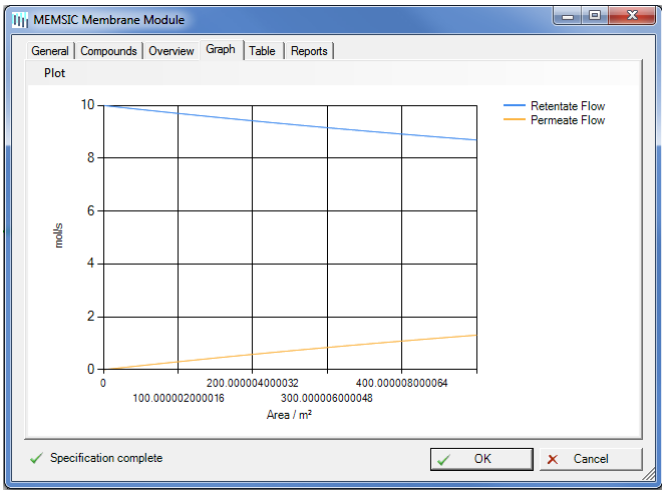
(Dual Mode)

How to use MEMSIC®

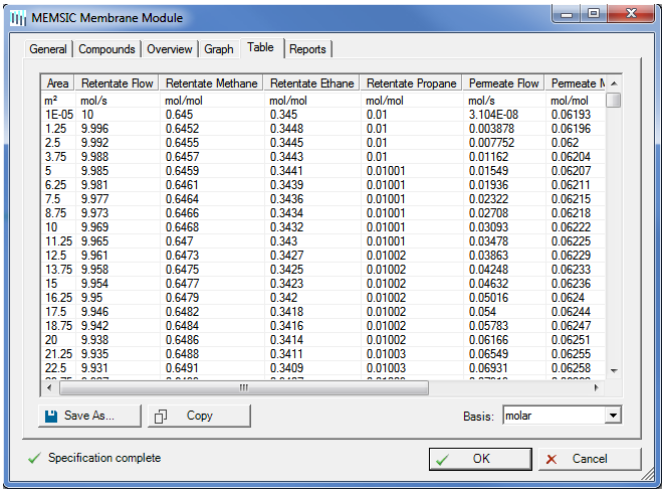
8. "Overview" / "Graph"/ "Table"/ "Reports": when the simulation have been run and the calculation is finished, those tabs give an overview of the results.



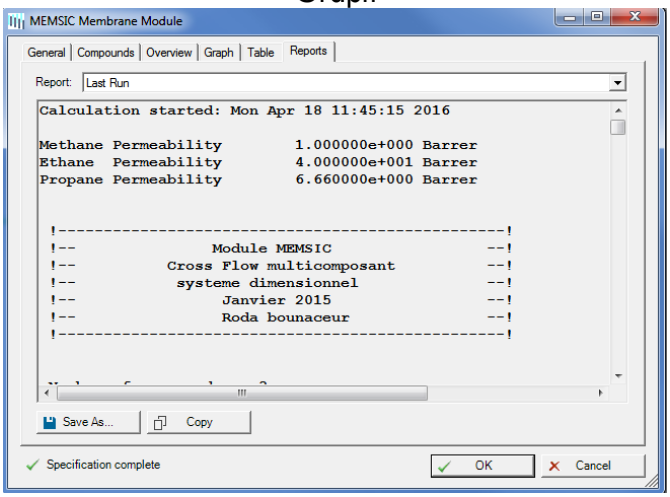
"Overview"



"Graph"



"Table"



"Reports"

9. By clicking on the "To Excel" button, all the results are stored in an Excel file.