Thermolib FAQ

1 Acquiring Thermolib

Q: I'M INTERESTED IN TRYING OUT THE THERMOLIB TOOLBOX. CAN I GET A TEST OR DEMO LICENSE?

A: Yes. Fill out the form on our website (http://www.eutech-scientific.de/products-services/tools/thermolib/thermolib-demo-request.html). You will receive an offer for test license that lasts in general 30 days from the day of activation. In this time, you will be able to try out all the features of a fully-licensed Thermolib.

Q: HOW MUCH DOES THERMOLIB COST?

A: Please send us a request for Thermolib. If you are working in academic research, make sure to mention this in your request so that we can offer you the academic discounts. For users from certain countries, we will redirect you to your respective reseller.

Q: I AM A STUDENT/PROFESSOR/WORK IN ACADEMIC RESEARCH. CAN I GET AN ACADEMIC DISCOUNT?

A: Yes. We have an attractive academic licensing model. Please send us a request for more information.

2 Requirements

Q: DO I NEED MATLAB/SIMULINK TO USE THERMOLIB?

A: Yes. At the time being, it is essential to have a running license of MATLAB/Simulink to use the Thermolib toolbox. If you have no Simulink license available, you can only use Thermolib command line functions like some basic thermodynamic state calculations and species diagram functions, however you cannot use the modeled components like pumps, compressors, heat exchangers, valves etc.

Q: WHAT ARE THE SOFTWARE AND HARDWARE REQUIREMENTS TO USE THE THERMOLIB TOOLBOX?

A:

Software:

Thermolib has been developed with MATLAB 7.5 (R2007b) and works with the following MATLAB versions (and later versions):

R2007b (32 bit)
R2008a (32 bit)

R2008b (32 bit)
R2009a (32 bit)
R2009b (32 bit, 64 bit)
R2010a (32 bit, 64 bit)
R2010b (32 bit, 64 bit)
R2011a (32 bit, 64 bit)
R2011b (32 bit, 64 bit)
R2012a (32 bit, 64 bit)

You will need a Windows operating system (XP, 7, Vista). If you have a different operating system and want to use Thermolib, please send us an e-mail to support@thermolib.de, that we can keep track of interest for other operating system support.

Optional: MS Excel is a spreadsheet program from Microsoft. Maintenance of species thermophysical data and use of balancing functionality in Thermolib is much easier if you have MS Excel installed. Thermolib uses Excel-File format .xls (MS-Excel 2003).

Hardware:

Hardware requirement depends on the licensing scheme you want to use. Thermolib can be licensed with software activation (binding to the hardware called CodeMeterAct) or with a USB Dongle (CodeMeter Dongle). For the latter you need a free USB port. Trial versions, we offer only with software activation.

Hard Disk Space:

A Thermolib installation requires approximately 120 MB on your hard disc.

3 INSTALLATION AND SETUP

Q: HOW DO I SETUP THERMOLIB IN MATLAB/SIMULINK?

Q: I FINISHED INSTALLING THERMOLIB BUT I CANNOT FIND THE TOOLBOX IN THE SIMULINK LIBRARY BROWSER.

A: After installation of Thermolib, the MATLAB environment has to be notified of the paths to the Thermolib toolbox. To do that, start the MATLAB version you wish to use Thermolib in, and go to the "Bin" folder of the Thermolib installation directory. Run "Thermolib_Setup" from the command line (step 11 in the Thermolib Installation Guide).

So as not to repeat the same procedure every time you open MATLAB, choose "Yes" when asked "Do you want to store the settings permanently?".

Q: Do I have to run the Thermolib setup every time I start MATLAB?

A: No. When you run "Thermolib_Setup" from the command line (step 11 in the Thermolib Installation Guide), choose "Yes" when asked "Do you want to store the settings permanently?".

If the settings are not being saved, it is possible that you do not have write-permission on your MATLAB installation. Please contact your IT administrator.

If for some reason it is not possible to save the settings, we suggest creating a MATLAB shortcut that will navigate to the Thermolib Bin directory and run the setup script. Consult the MATLAB help by searching for "shortcut".

Q: Can I use multiple Thermolib versions in parallel on one computer?

A: Yes. To this end, we recommend to not permanently store the path settings in your MATLAB installation, but rather create a "shortcut" for each Thermolib version, which you want to use. After starting MATLAB you choose the shortcut of the version, which you want to use today. The shortcut should call the Thermolib_Setup-script of the corresponding Thermolib installation and could look like the following:

```
cd('c:\Program Files\EUtech SE\Thermolib\5.1.1\Bin');
Thermolib_Setup('ShowDialog','off','SavePaths','off')
cd ..
```

The additional arguments for Thermolib_Setup avoid the pop-up dialog asking the "save path question" and by default do not save the path settings.

4   Species and Media

Q: Which compounds/species are available as part of the Thermolib chemical media database?

A: Thermolib is currently with data for the following species. Additional species can be added (Please refer to chapter 4 "Chemical Media Database" in the Thermolib user manual.)

- CH4
- H2O
- H2O-IF97
- CO
- CO2
- H2
- N2
- O2
- isooctane
- Methanol
- 1-Buten
- 1,3-Butadien

- R134a, no chem.
- NH3
- NO
- Ethanol
- n-Propane
- n-Butane
- i-Pentane
- R12
- R407C, no chem
- R125, no chem
- R32, no chem
- R1234yf, no chem.

Q: CAN I ADD NEW COMPOUNDS/SPECIES TO THE THERMOLIB CHEMICAL MEDIA DATABASE? HOW?

A: Yes. You can adjust the Thermolib chemical media database by adding either new properties or new species, or both. One feature of Thermolib is that the chemical media data can be viewed and adjusted in Microsoft Excel. Thermolib offers a tool where it automatically imports the chemical media data from Excel into a structure SMediaData into the MATLAB workspace. If you do not have Microsoft Excel available, you can adjust the SMediaData structure in MATLAB directly.

Please refer to chapter 4 "Chemical Media Database" in the Thermolib user manual. There you will find specific instruction on how to add a new species and new properties (section 4.3), and where to find the thermo-physical properties for new compounds (section 4.5).

Q: WHICH THERMODYNAMIC VALUES/PROPERTIES DOES THERMOLIB COMPUTE?

A: Thermolib describes the medium flow between blocks using a flow bus. The bus contains a set of the most important properties defining the flow:

- molar flow rate (ndot),
- pressure (p),
- temperature (T),
- enthalpy flow (Hdot),
- entropy flow (Sdot),
- flow of the Gibbs free energy (Gdot),
- capacity flow (Cpdot=ndot*cpm),
- vapor fractions (x) and the
- molar species concentrations (psi).

See section 1.3.3 in the Thermolib user manual.

Q: CAN I MODEL SOLID STATES IN THERMOLIB?

A: No. Thermolib models mainly liquid and gas flows. The exception is solid compounds that behave similar to gas, for instance pulverized coal or dust in a gas flow. It is possible to model these in Thermolib with some restrictions.

As a workaround to model solid state of dust, solid fuels or ash in gas flows, we recommend to model the solid species as "liquid", as typically in such flows no liquid state appears. That means the "x=0" on the Thermolib flow bus changes meaning from "liquid" to "solid". To this end, the vapor-pressure curve for this species has to be manipulated accordingly. The species properties like "liquid density" then represent the solid property instead. The "Gas Dryer" component can be used as a "Filter" to separate the solid dust/ash/coal from the gas flow.

Q: CAN I MODEL LIQUID/GAS STATES IN THERMOLIB?

A: Yes. Thermolib also covers 2-phase region and supercritical states.

Q: CAN THERMOLIB HANDLE MIXTURES?

A: Yes. Thermolib can handle mixtures. E. g. wet air is modeled by mixing the compounds oxygen, nitrogen and water. Especially combustion products and refrigerant mixtures can be modeled. However the thermodynamic mixture model in Thermolib does not handle interactions between the species and aceotropic effects.

5 MODELING IN SIMULINK

Q: IN WHICH RANGE, FOR BOTH TEMPERATURE AND PRESSURE, DOES THERMOLIB SIMULATE?

A: The global range for temperature is between 1 and 6000 Kelvin. The global range for pressure is between 1 Pa and 1000 bar. The valid range in a Thermolib model, however, is dependent on the species being used in that model.

Q: CAN I MODEL PRESSURE FEEDBACK IN A HYDRAULIC NETWORK USING THERMOLIB?

A: Yes. Please refer to chapter 5 in the user manual (\Help\Thermolib-UserManual.pdf) for further help on pressure feedback in Thermolib.

Q: HOW CAN I MODEL A CLOSED LOOP SYSTEM USING THERMOLIB?

A: There are two options to build a closed loop with Thermolib: either use an initial-conditions block, or make sure there a buffer (a tank) in the loop. See the Demo Loops under Demos/Simulink/Thermodynamic Systems Library/ Pressure Feedback or the Carnot Cycle under Demos/Simulink/Thermodynamic Systems Library/ Basic Thermodynamic Cycle.

However, keep in mind that an initial-condition block is only a set of memory blocks for the elements of the Flow Bus. If you create cycles with initial condition blocks make sure, that you do sufficiently fix the conditions of the cycle to avoid unintended "dynamics".

Q: WHICH THERMODYNAMIC EQUATIONS OF STATE ARE USED?

A: You have a choice between "Ideal Gas" and "Real Gas (Peng-Robinson)" equations of state. In the Thermolib model setup block, you can choose which equations of states your model should use for modeling the gas phase.

Q: CAN I MODEL DYNAMIC PROCESSES WITH THERMOLIB?

A: Yes. Most Thermolib components contain thermal masses, which mainly govern the dynamic behavior of a thermodynamic process. Furthermore you can use a generic tank block to model mass storage behavior.

Q: WHAT CAN I DO IF THE COMPONENT I NEED IS NOT PART OF THE THERMOLIB LIBRARY?

A: Thermolib offers basic thermodynamic building blocks that simplify building new components. These building blocks take care of calculating the thermodynamic states and properties and make sure that all thermodynamic calculations are correct.

You can use these basic building blocks to build more complex components. A good starting point is to take a look how some of the Thermolib components themselves are built and modifying them accordingly.

Q: I WOULD LIKE TO MODIFY A THERMOLIB COMPONENT OR CREATE A NEW ONE. CAN I DO THAT?

A: Yes. All the Simulink code in Thermolib is visible for users and can be modified as they see fit. Only the code for the basic building blocks (Thermodynamic States and Properties) is not visible and cannot be modified. The Simulink code constitutes a large part of the Thermolib library and is generally a good starting point to modify and create new blocks.

Q: IS IT POSSIBLE TO VIEW/MODIFY THERMOLIB SOURCE CODE?

A: Yes and no. Please see the answer for the question "I would like to modify a Thermolib component or create a new one. Can I do that?".

Q: WHAT ARE THE RECOMMENDED SOLVER SETTING FOR THERMOLIB IN SIMULINK?

A: There is no general solver setting recommendation for Thermolib models as it strongly depends on the problem.

6   TROUBLESHOOTING

Q: I GET A "CERTIFIED TIME ERROR /LICENSE ERROR" WHEN I TRY TO RUN THERMOLIB. WHAT CAN I DO?

A: These are the steps to update a certified time on a CodeMeter USB-stick (e.g. for Thermolib licenses):

1. Open the CodeMeter Control Center from your taskbar by double clicking on the CodeMeter icon.

2. Start the WebAdmin interface from the CodeMeter Control Center by clicking on the button "WebAdmin".

3. Now your web browser should open and you can navigate to "Content => CM-Stick" to initiate a certified time update.

See section 2 in the Thermolib-LicenseUpdate.pdf help file in the \Help folder of your Thermolib installation. You have to make sure, that the certified timestamp is not older than 90 days. The days remaining are displayed by the Thermolib_Version command.

Q: I HAVE AN ALGEBRAIC LOOP IN MY THERMOLIB MODEL. WHAT CAN I DO TO ELIMINATE IT?

A: See the answer for the question "How can I model a closed loop system using Thermolib".

Q: MY THERMOLIB MODEL IS VERY SLOW. IS THERE ANYTHING I CAN DO TO SPEED IT UP?

A: Thermolib provides a convenient way of displaying the content of the FlowBus signals using the FlowDisplay block. Unfortunately this visualization slows down the simulation time. To disable the update of the FlowDisplay blocks, create a base workspace variable "th_disable_flow_display". If this variable is present, the FlowDisplay blocks will not be updated any more.

Second typical source of slow down is fast dynamics. Check if you modeled very small volumes compared to big volume flows of media. This might lead the solver to decrease the simulation step size. Consider simplifying the model.