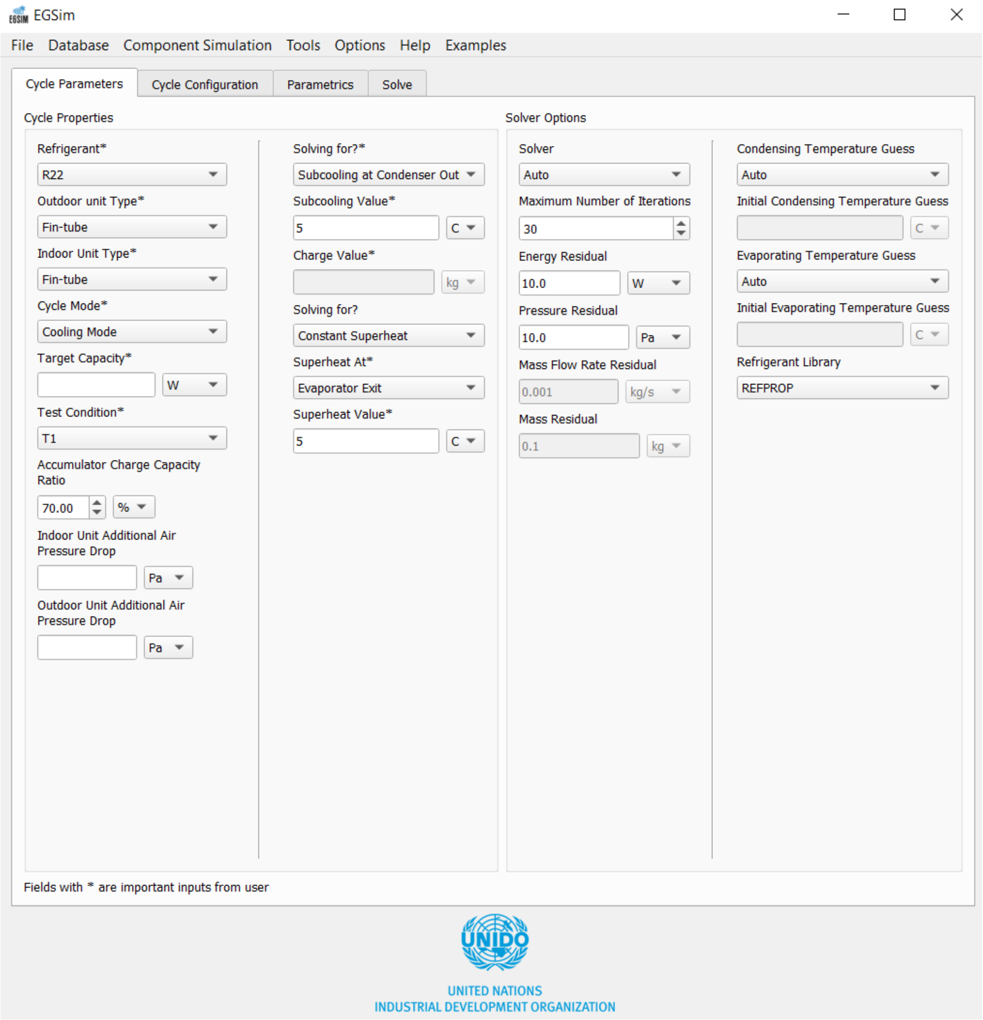
Catalog

1. Introduction

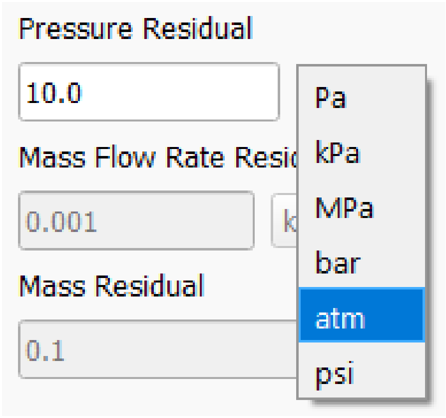
EGSim User Guide

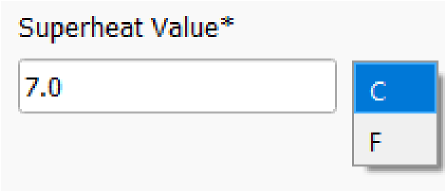
EGSim

- Simulates vapor compression cycles (VCC) designed using air-to-refrigerant heat exchangers
- Physics-based modeling
  - Inputs: components physical information
  - Outputs: performance parameters including COP, capacity, power and charge
- Built-in modeling support intelligence
- Accurate
- Friendly user interface

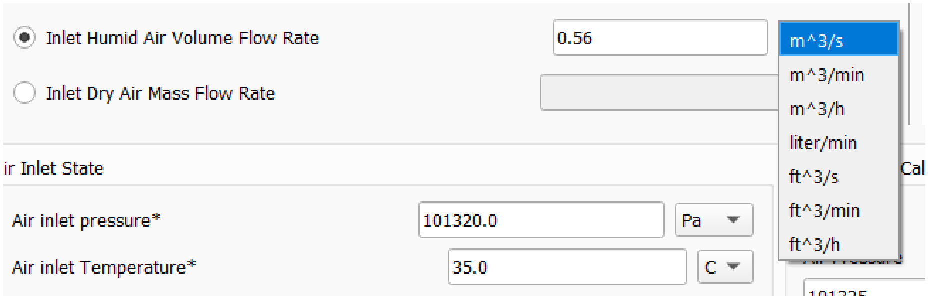


Unit Systems

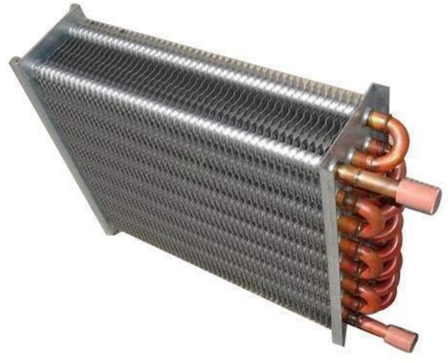




EGSim is equipped with unit system management that is convenient for both English and SI units

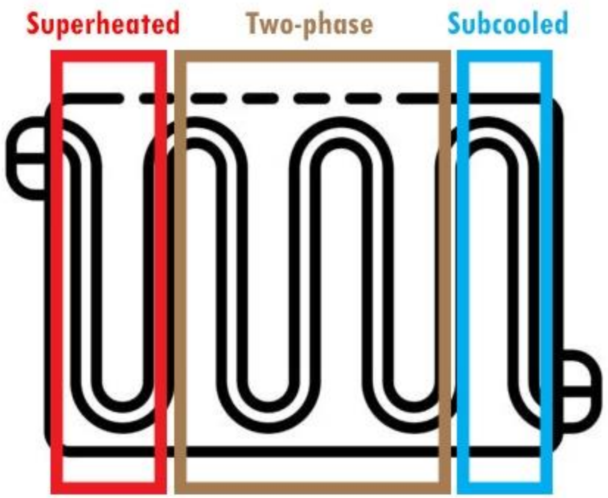


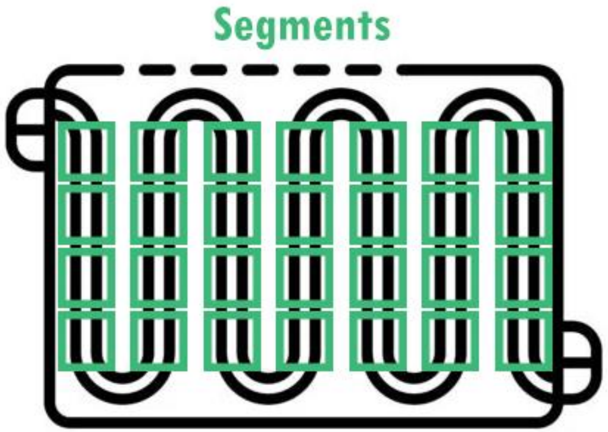
Heat Exchangers



Used to simulate the evaporator and the condenser

- Fin-tube or microchannel HX
- Phase-by-phase or segmented

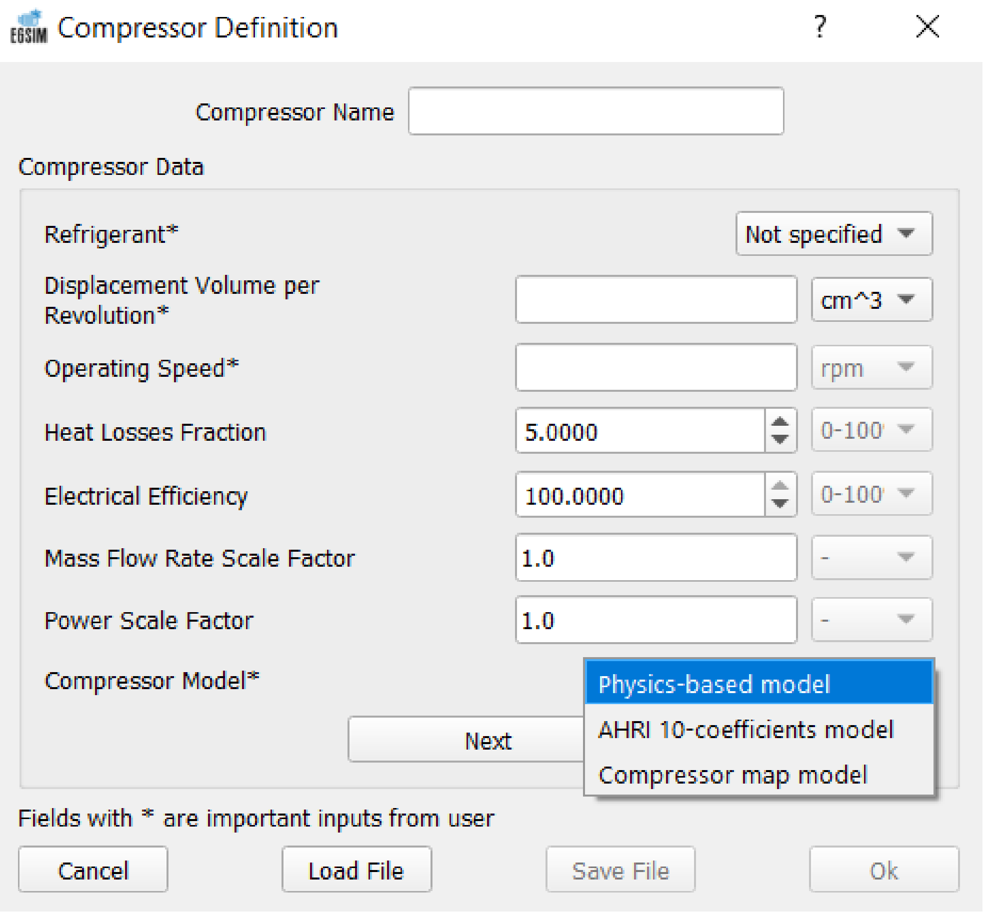




Compressors

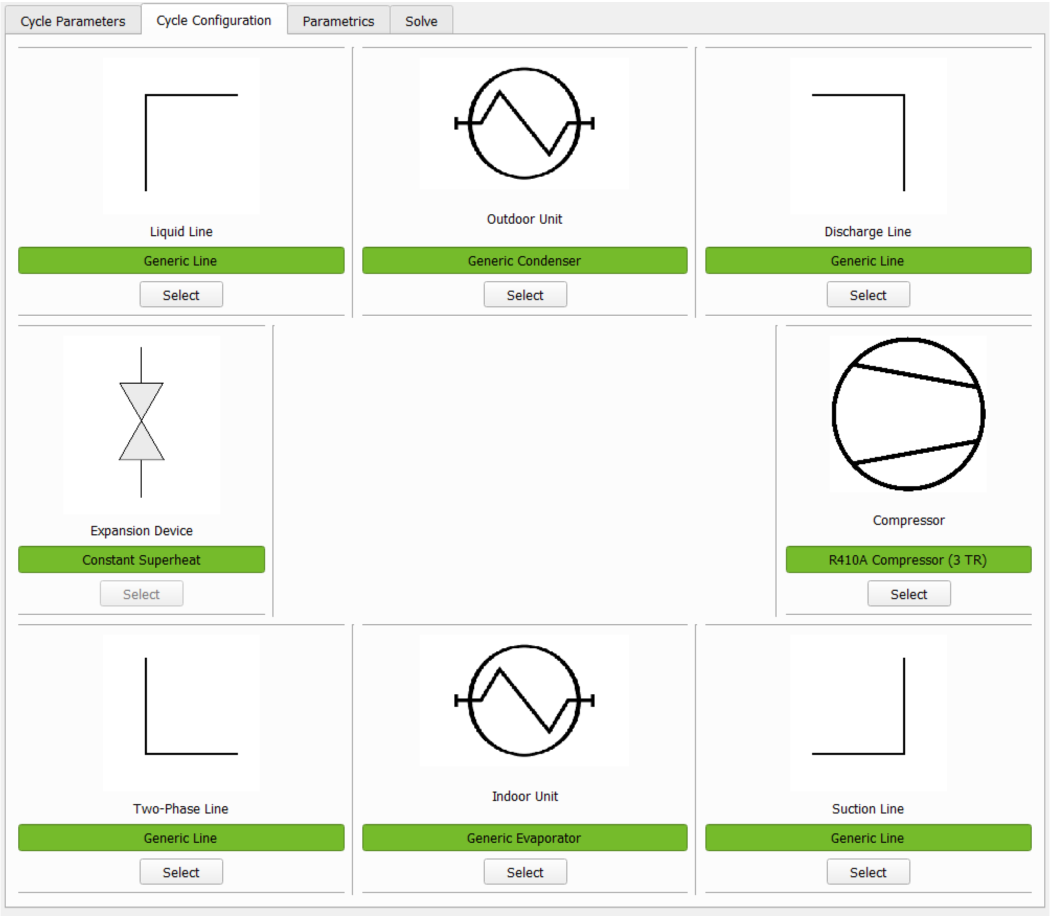
- Physics-based Model
  - Isentropic efficiency*
  - Volumetric efficiency*

  * Efficiency can be defined as a function of pressure ratio
- Performance maps with a capability of simulating variable speed compressors
- ARI-210/240 10 Coefficient Model with a capability of simulating variable speed compressors
- Built-in compressor database for major OEMs



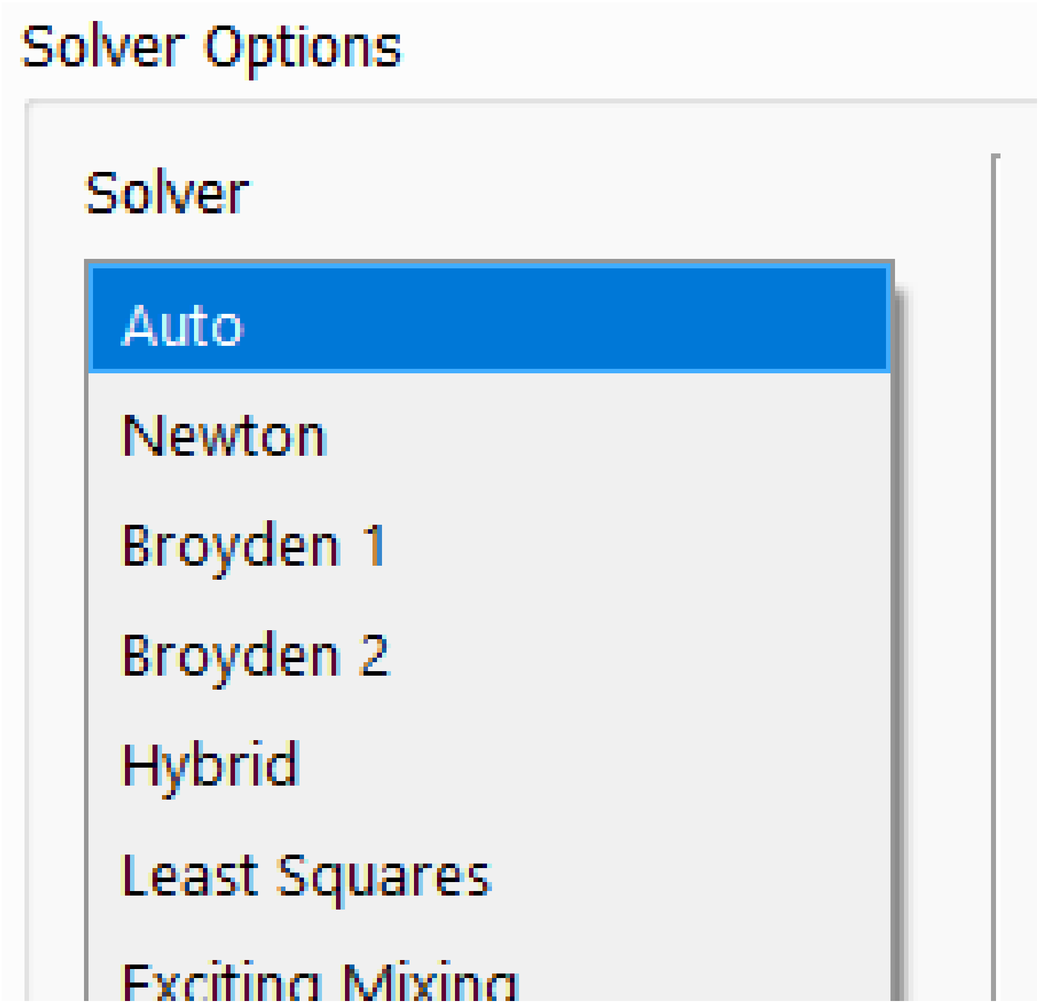
Connecting Lines

- Lines connecting the components are also simulated for heat transfer and pressure drop, along with charge calculations
  - Discharge line: Compressor to condenser
  - Condenser to expansion device (capillary tube)
  - Expansion device (capillary tube) to evaporator
  - Suction line: evaporator to compressor



Cycle Solvers

- Several solvers can be used for the cycle to improve simulation stability
- Default solvers:
    - Least Squares: constant superheat at compressor inlet
    - Newton: constant superheat at evaporator exit



Detailed Results

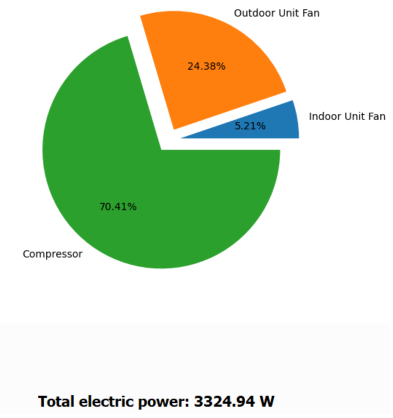
- Detailed cycle results with all performance data
- Detailed results for each component
  - Inlet and outlet conditions for all fluid streams (refrigerant and air)
  - Performance assessment characteristics
  - Relevant input data

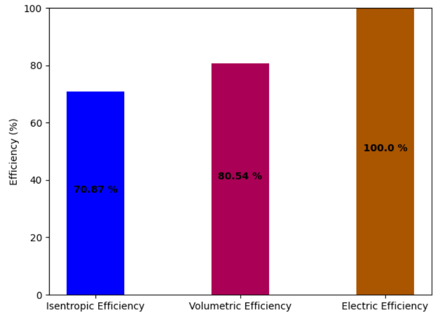
Cycle | Compressor | Outdoor Unit | Indoor Unit | Liquid Line | Two-phase Line | Suction Line | Discl

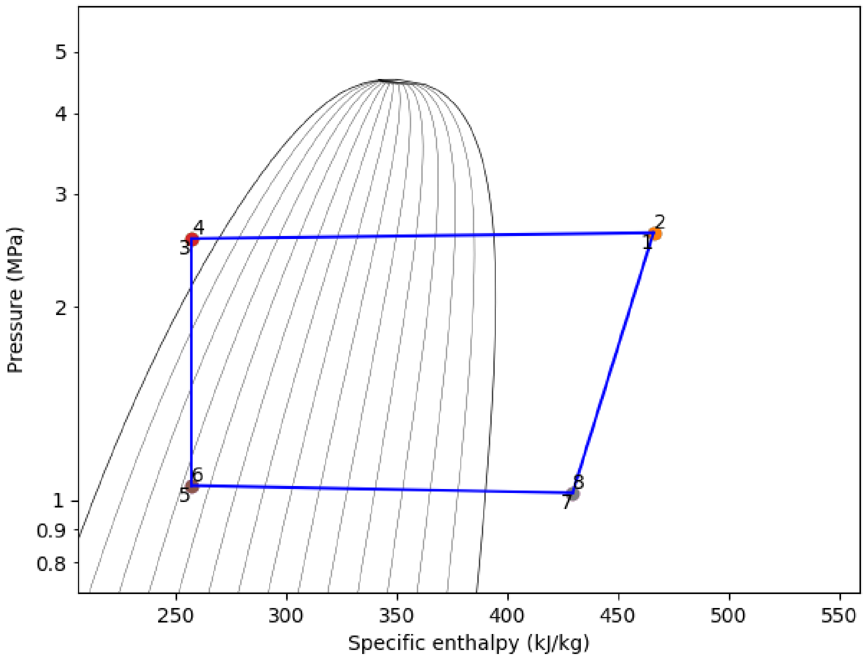
	Parameter	Unit	Value
1	Name		R410A Compressor (3 TR)
2			
3	Compressor Speed	rpm	2500
4	Displacement volume per revolution	m^3	5e-05
5	Heat Loss Fraction	%	0
6	Power scale factor	-	1
7	Mass flow rate scale factor	-	1
8			
9			
10	Mechanical power	W	2341.1
11	Electrical power	W	2341.1
12			
13	Mass flow rate	kg/s	0.063921
14			
15	Inlet Pressure	Pa	1.0264e+06
16	Inlet Temperature	C	13.141
17	Inlet Enthalpy	J/kg	4.2949e+05
18	Inlet Entropy	J/kg.K	1815.7
19			
20	Outlet Pressure	Pa	2.6082e+06
21	Outlet Temperature	C	71.004
22	Outlet Enthalpy	J/kg	4.6611e+05
23	Outlet Entropy	J/kg.K	1847.1

Results' Visualization

- Visual plots and graphs are shown in the results for comprehensive analysis

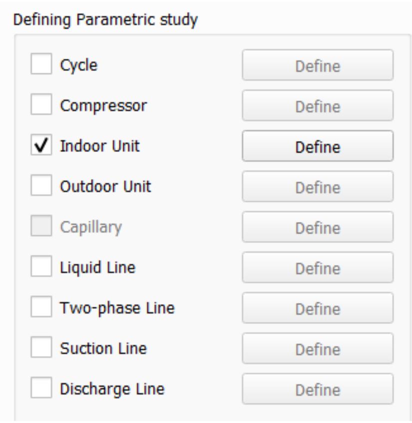


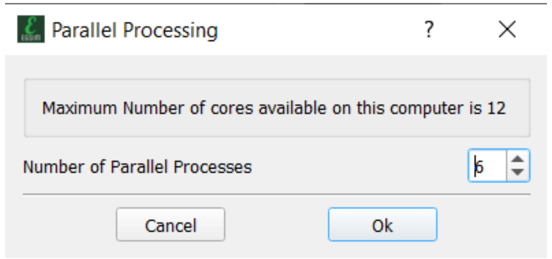


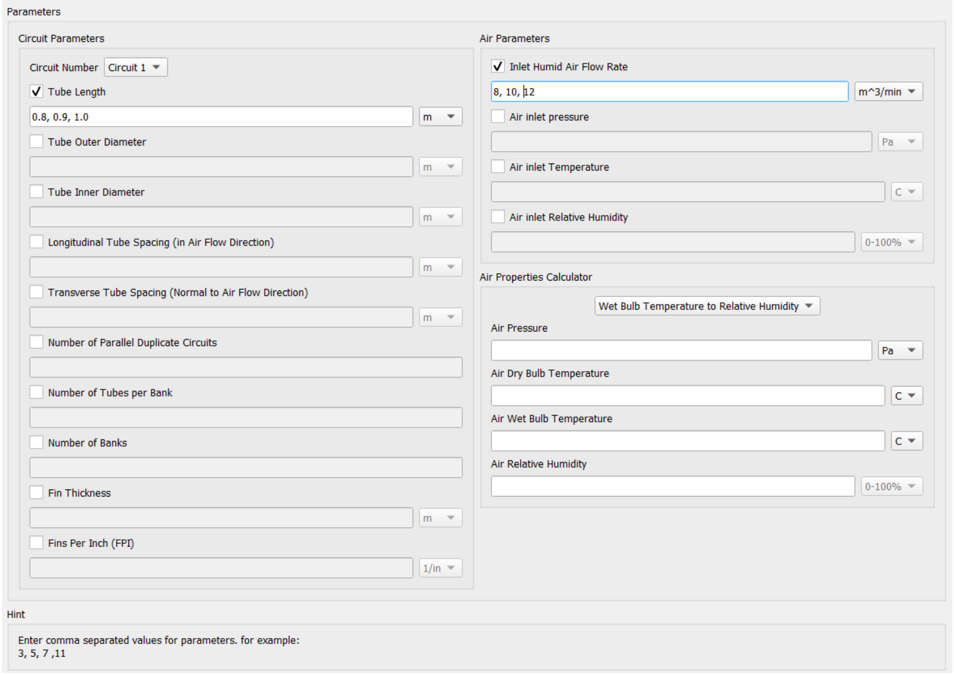


Parametric Study

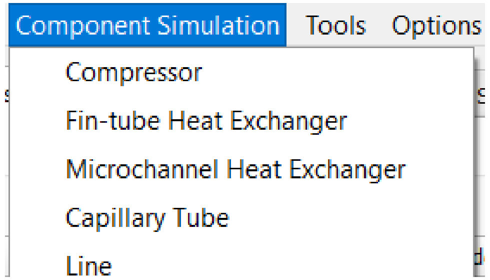
- The ability to perform a parametric study on any design variable of any component with parallel processing capability



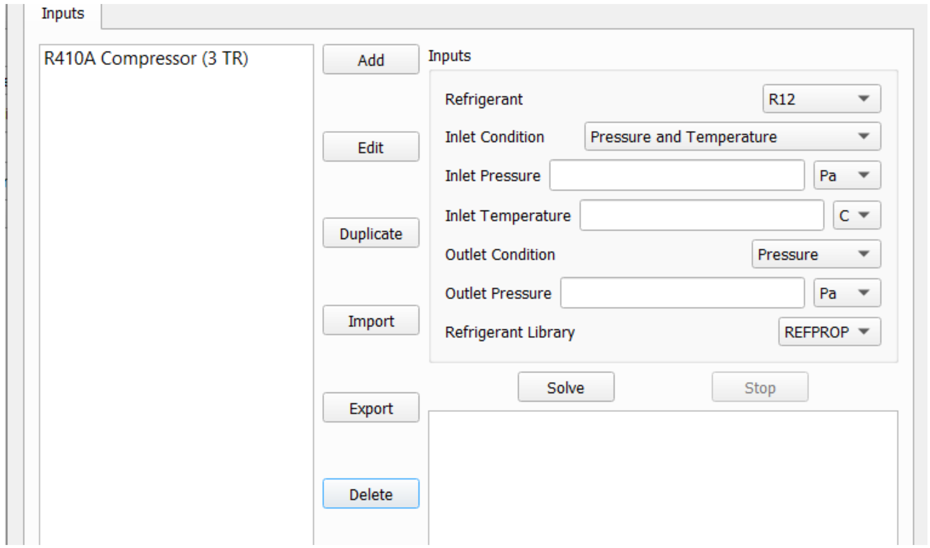




Component Simulation

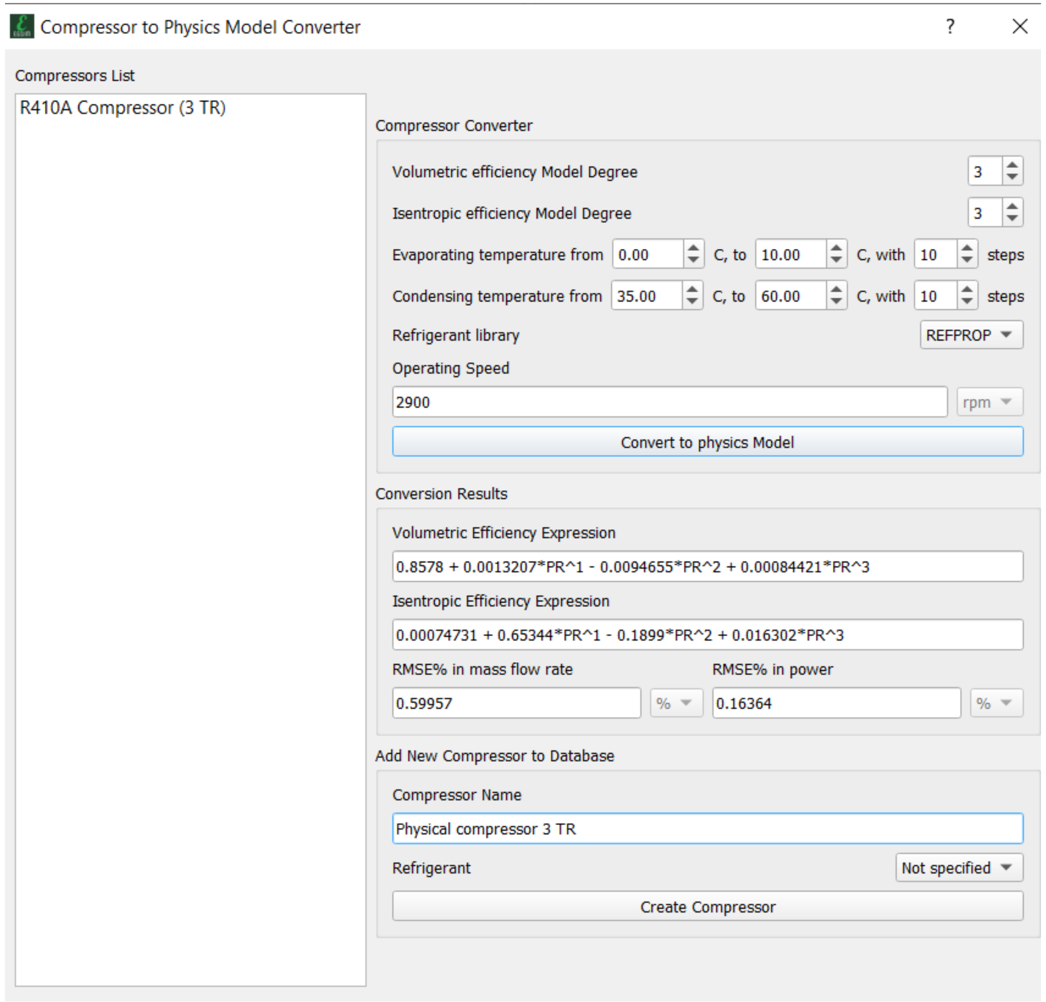


- The ability to simulate any cycle component individually for any inlet condition



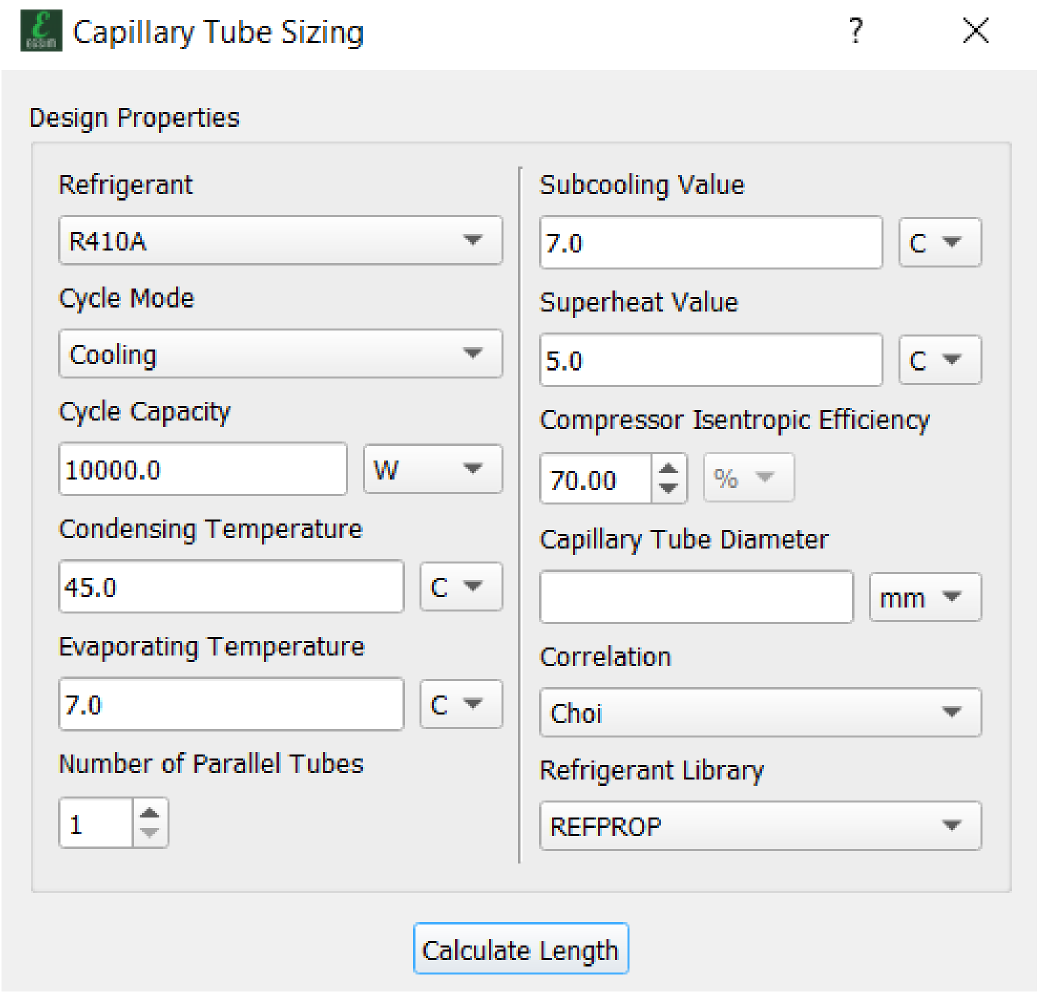
Compressor Model Converter

- The ability to convert a compressor curves model to physics-based model
- Develop isentropic and volumetric efficiencies (polynomial function) using available data from baseline refrigerant
- Assume that the efficiencies will remain the same for other refrigerants



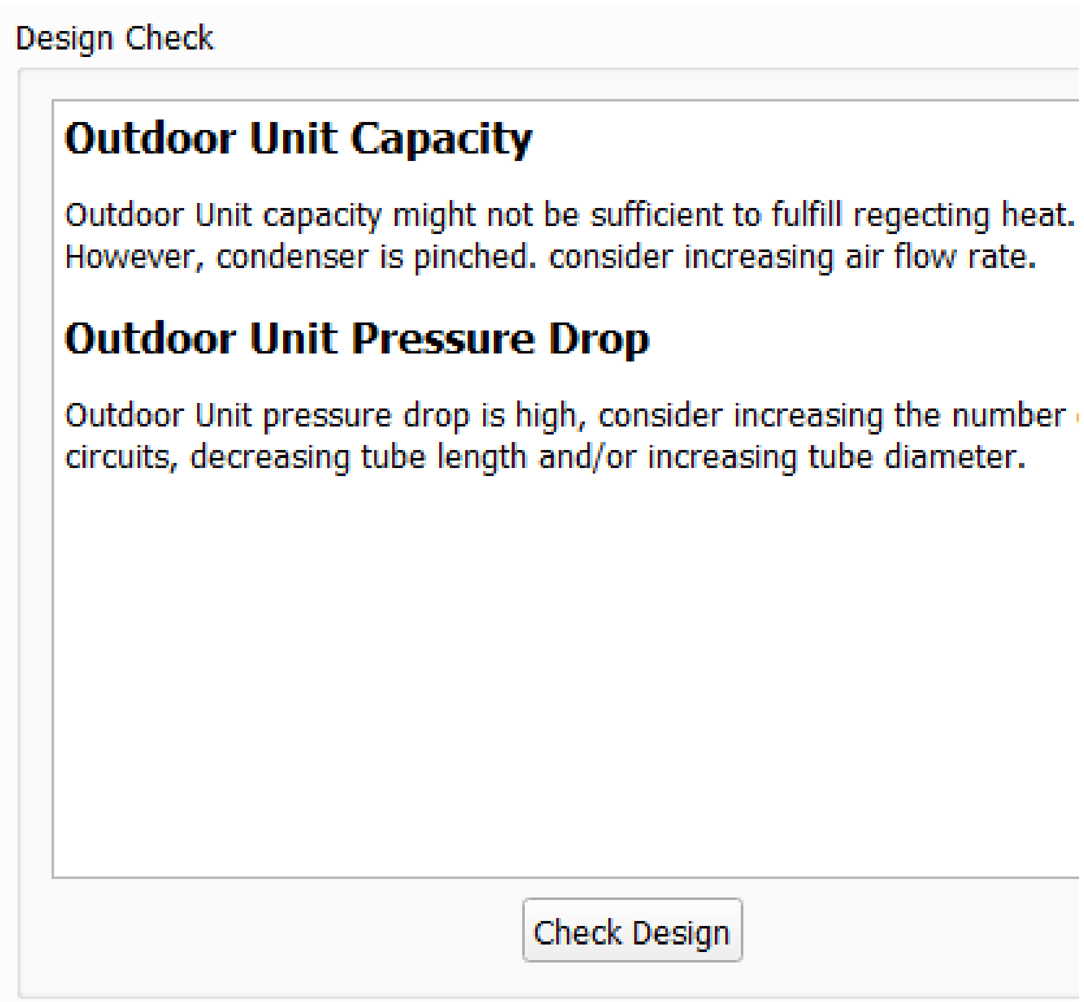
Capillary Sizing

- The capillary sizing tool is based
  - Target capacity
  - Cycle superheat
  - Cycle subcooling
  - Saturation temperatures
  - Compressor isentropic efficiency



Design Check

- Check condenser, evaporator and compressor ability to
  - meet the target capacity
  - Maintain reasonable pressure drop in heat exchangers
- Provide recommendations for the modifications that can be implemented to the component(s) in case there is a predicted design issue



Exporting Results to Excel

- Using the export wizard, results can be safely stored in an excel file to perform further analysis if needed

	A	B	C	D	E
1		**Parameter**	**Unit**	**Value**	
2	**0**	Solved?		Yes	
3	**1**	Solver Error			
4	**2**				
5	**3**	Refrigerant		R410A	
6	**4**				
7	**5**	Cycle Capacity	W	10848.523443659977	
8	**6**	Cycle Power	W	3324.943357682919	
9	**7**	Cycle Charge	kg	4.907157208414286	
10	**8**				
11	**9**	Cycle Condensing Temperature	C	43.13606492571023	
12	**10**	Cycle Evaporating Temperature	C	8.140507742085276	
13	**11**				
14	**12**	Cycle COP	W/W	4.707832340898341	
15	**13**	System COP	W/W	3.2627693998432736	
16	**14**				
17	**15**	Refrigerant Mass Flow Rate	kg/s	0.06392143396410277	
18	**16**				
19	**17**	Outdoor Unit Subcooling	K	7.005495227867982	
20	**18**	Indoor Unit Superheat	K	5.008718701813848	
21	**19**	Compressor Superheat	K	5.000000000000057	
22	**20**				
23	**21**	Outdoor Unit TTD	K	1.7083436191750252	
24	**22**	Indoor Unit TTD	K	2.790687363968914	
25	**23**				
26	**24**	Energy Balance	W	0.6355801110075845	
27	**25**	Expansion Device	-	TXV	

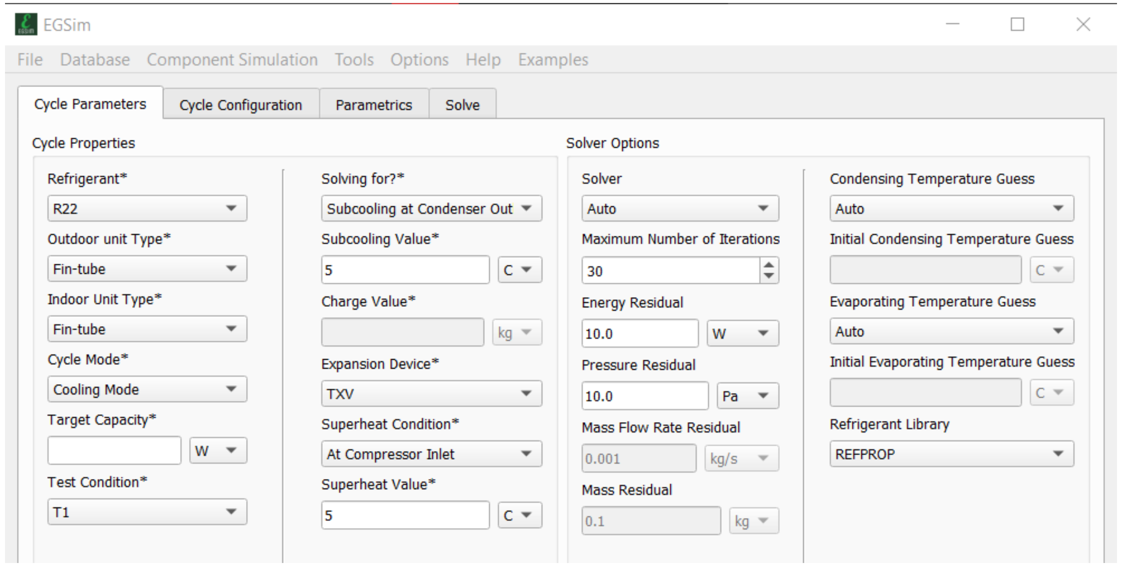
Cycle | Compressor | Outdoor Unit | Outdoor Unit Fan | Outdoor Unit Ci

2. Cycle Parameters

EGSim User Guide

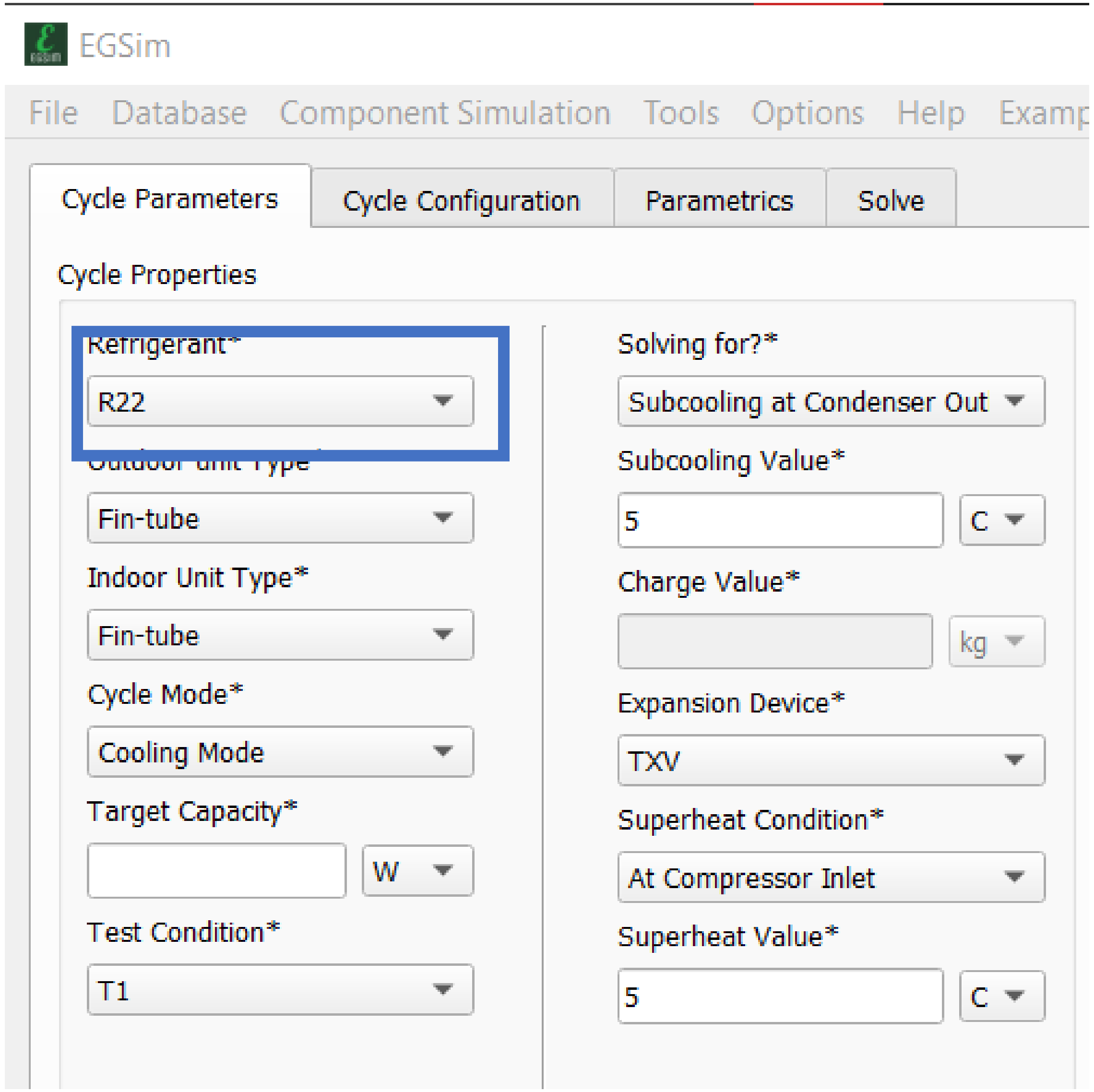
Overview

- This chapter describes the cycle parameters tab
- This tab is considered as the landing tab where we define the main cycle parameters and the solver options



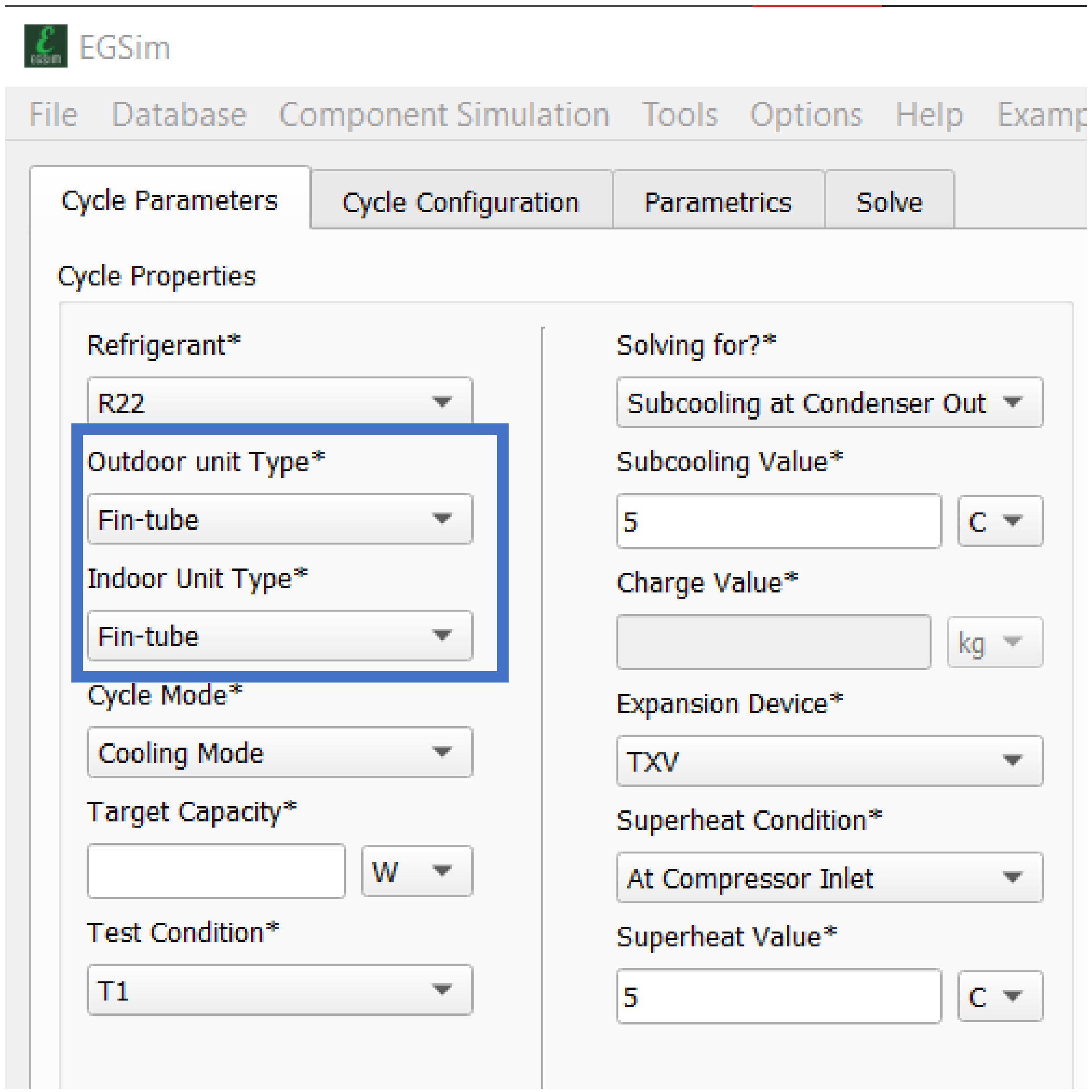
Refrigerant

- Using the dropdown menu for the refrigerant, we can select the main refrigerant used in our system
- Currently, EGSim supports 34 refrigerants:
  - R12
  - R123
  - R1233zd(E)
  - R1234yf
  - R1234ze(E)
  - R1234ze(Z)
  - R125
  - R134a
  - R143a
  - R152A
  - R22
  - R23
  - R32
  - R404A
  - R407A
  - R407B
  - R407C
  - R407D
  - R407E
  - R407F
  - R407G
  - R410A
  - R444B
  - R448A
  - R449A
  - R449B
  - R452A
  - R452B
  - R452C
  - R454A
  - R454B
  - R454C
  - R507A
  - R513A



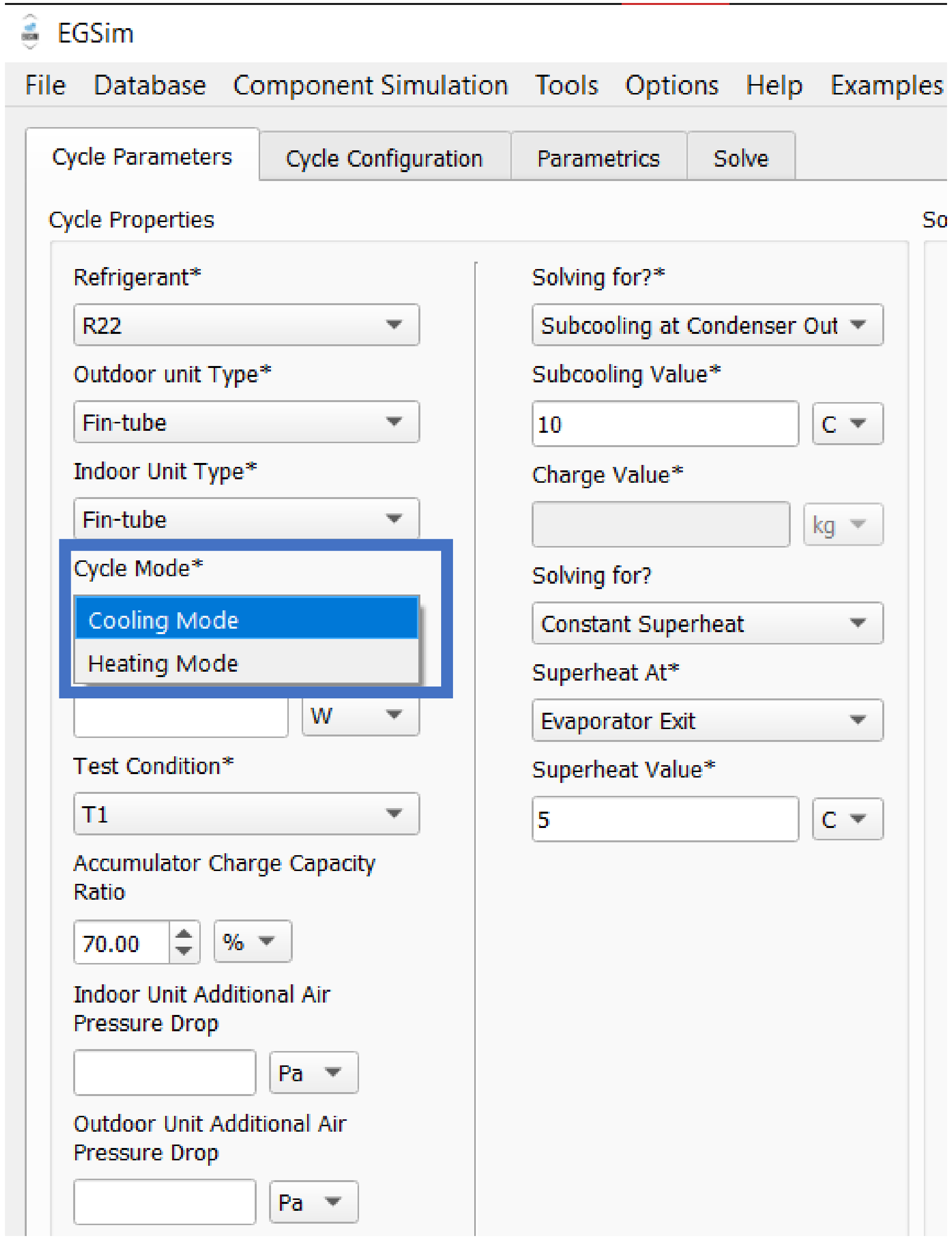
Outdoor Unit and Indoor Unit Types

- Using the dropdown list, the user can select the heat exchanger type (fin-tube or Microchannel) used for the indoor and outdoor units



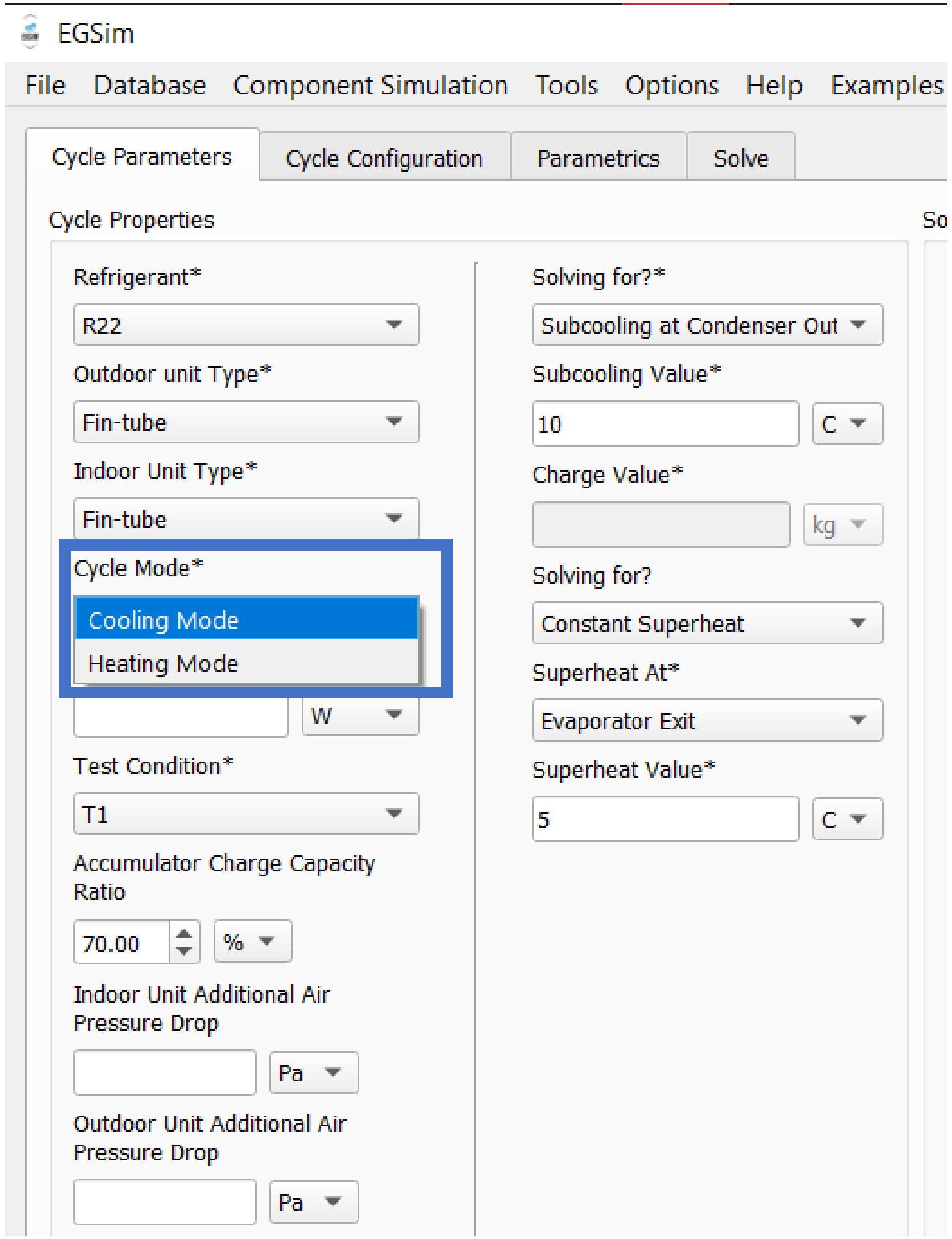
Cycle Mode

- Using the dropdown list, the user can select the cycle mode:
  - **Cooling mode:** this will treat the indoor unit as the evaporator, and the outdoor unit as the condenser
  - **Heating mode:** this will treat the indoor unit as the condenser, and the outdoor unit as the evaporator



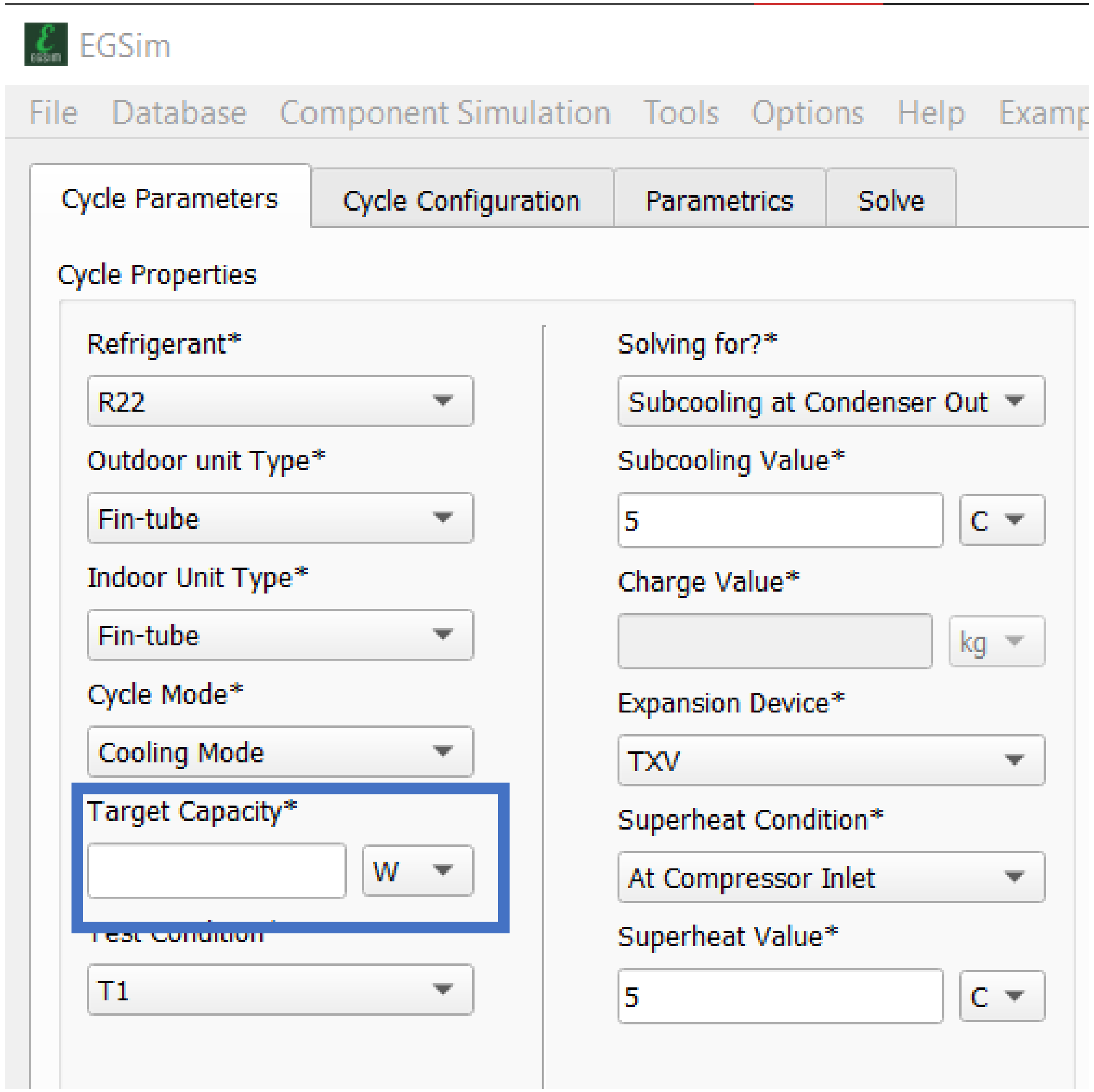
Cycle Mode, cont.

- System performance parameters depend on the cycle mode (e.g. capacity and COP)
- Cycle mode will affect the available test condition options
- EGSim will handle corresponding changes in components order while changing cycle mode from cooling to heating or vice versa
- Changing Cycle mode might require the user to change heat exchangers' circuit definition



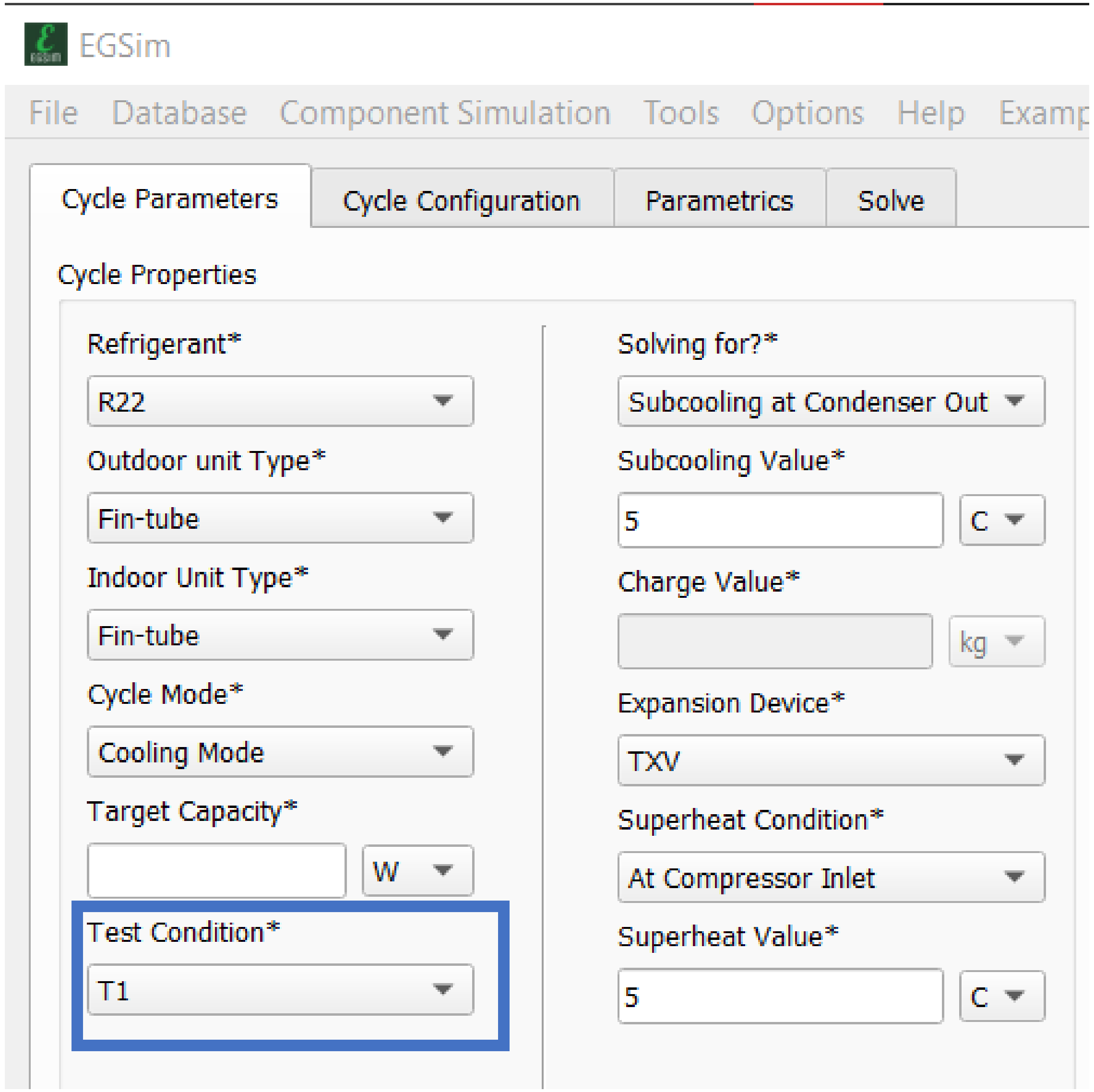
Target Capacity

- In this field, the user inputs his/her target system capacity
- The target capacity is the capacity from indoor unit whether it is operating in the cooling or the heating mode
- Target Capacity will be used in the design check features, but will not affect the cycle simulation/solver



Test Condition

- Test condition can be either
  - An ISO 5151 testing condition
  - A “custom” testing condition
- If an ISO 5151 testing condition is chosen, the defined air inlet condition for the heat exchangers will be ignored
- If “custom” testing condition is chosen, the defined air inlet condition in heat exchangers will be considered



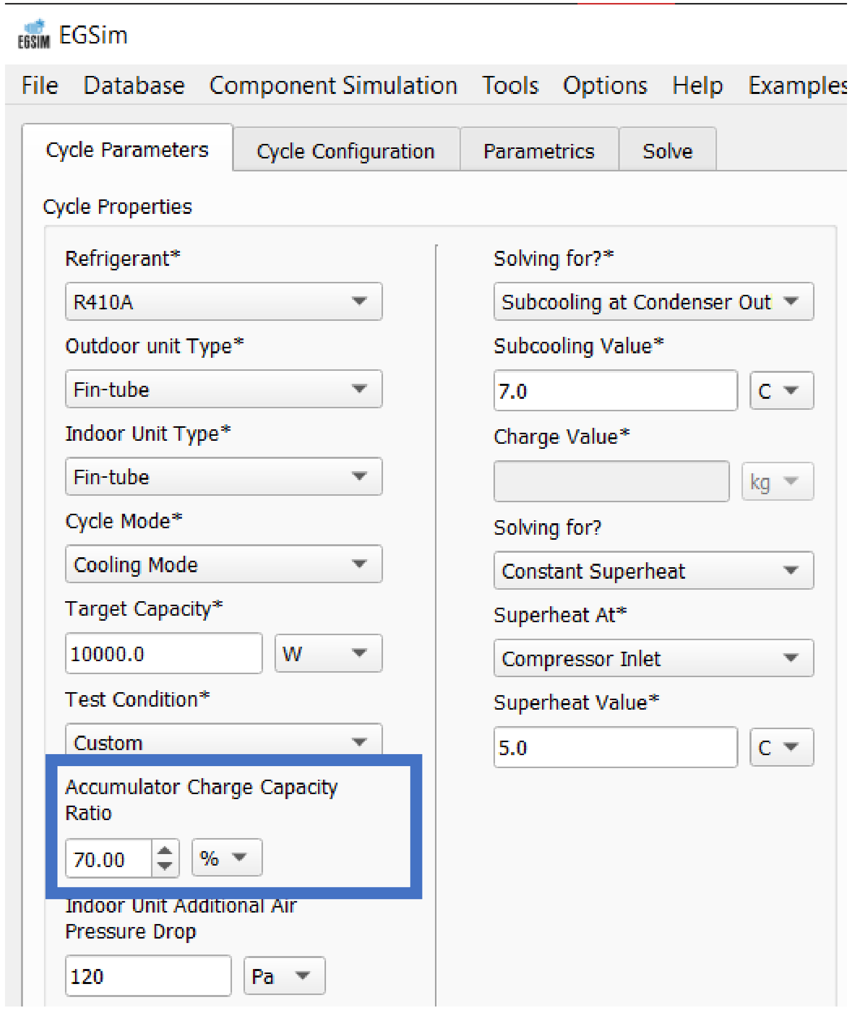
Test Condition, cont.

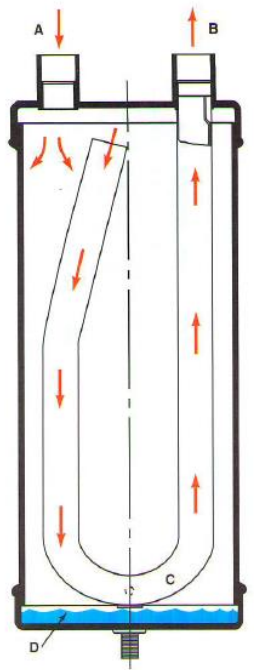
The following table shows the ISO 5151 testing conditions

		Indoor DBT (˚C)	Indoor WBT (˚C)	Outdoor DBT (˚C)	Outdoor WBT (˚C)
Cooling	T1	27	19	35	24
	T2	21	15	27	19
	T3	29	19	46	24
Heating	H1	20	15	7	6
	H2	20	15	2	1
	H3	20	15	-7	-8

Accumulator Capacity Ratio

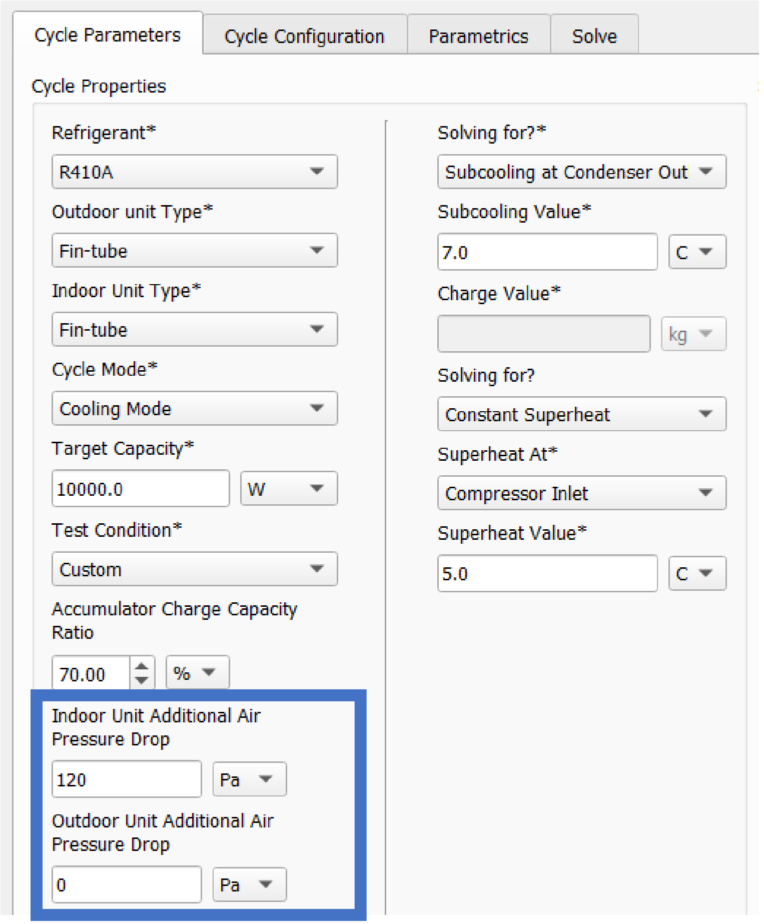
- Accumulator capacity ratio is the ratio of refrigerant charge stored in the accumulator to the total system charge
- The default value is 70%





Additional Air Pressure Drop

- For indoor and outdoor units, additional air-side pressure drop might be needed to account for throw or additional components other than heat exchangers
- You can define each unit's additional air-side pressure drop, which will be added to the pressure used to estimate the unit fan performance



How The EGSim Solver Works?

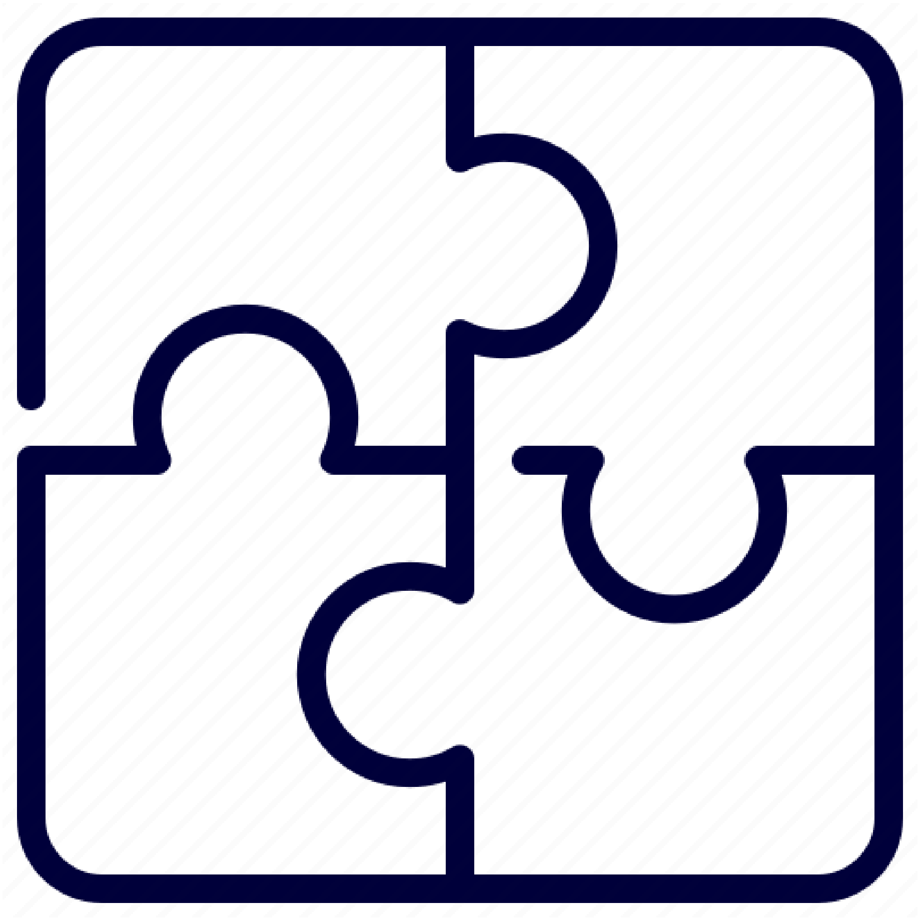
- The solver takes two inputs:
    - Input 1: can be either:
        - Subcooling of condenser
        - Cycle charge
    - Input 2: Expansion device which can be:
        - Defined by a superheat value at:
            - Evaporator exit
            - Compressor inlet
        - Capillary tube
- The solver then tries to find the evaporating and condensing temperatures that will achieve a target energy balance for the cycle

How The EGSim Solver Works?, cont.

- In case the condenser subcooling is selected, the solver will consider an energy residual consisting of the error in the calculated the subcooling value
- In case the system charge is selected, the solver will consider a mass residual consisting of the error in the calculated system charge

How The EGSim Solver Works?, cont.

- When constant superheat is selected, the solver will consider an energy residual consisting of the error in the calculated superheat value
- When the capillary tube model is selected, the solver will consider a mass flowrate residual consisting of the error in the calculated mass flowrate between the capillary tube and the compressor

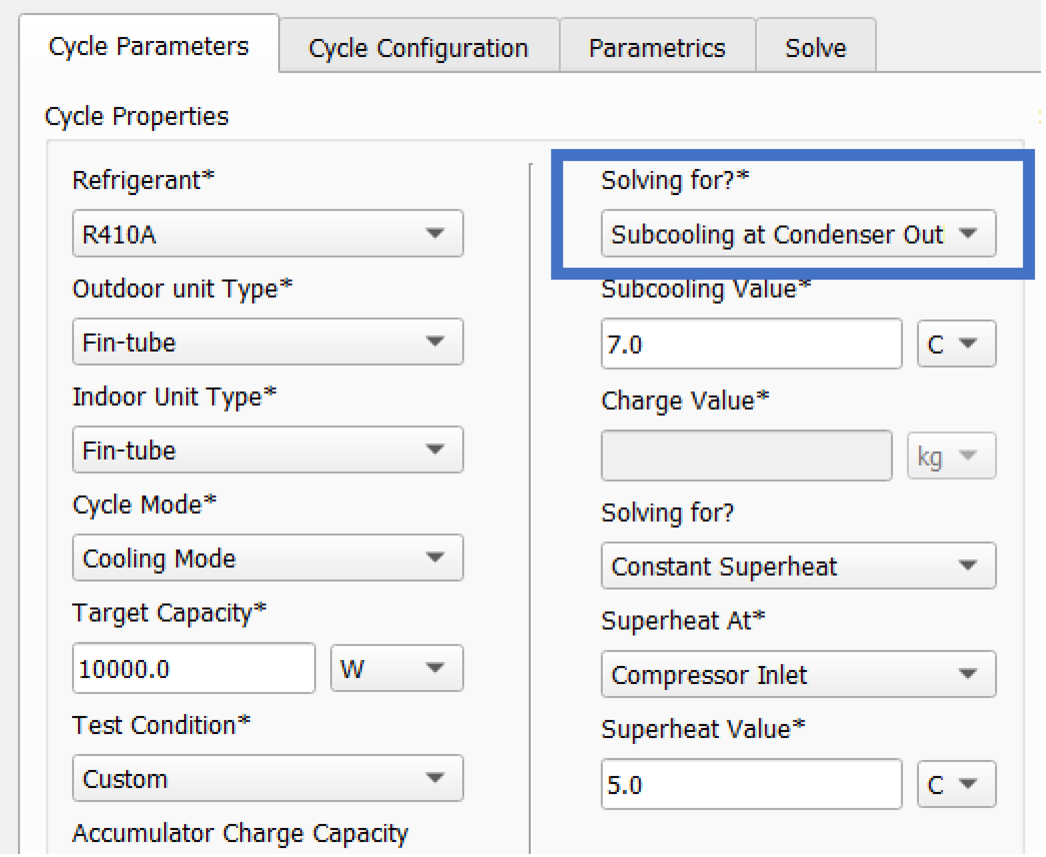


EGSim Solver Sequence

- The solution process is divided into inner and outer loops
- In the inner loop, the solver solve for the two system inputs using the corresponding residual functions
- Once the residual criteria is reached, the solver moves to the outer loop to calculate the sum of refrigerant side pressure drop in each component
- The solver moves back to the inner loop if the pressure convergence criteria is not met

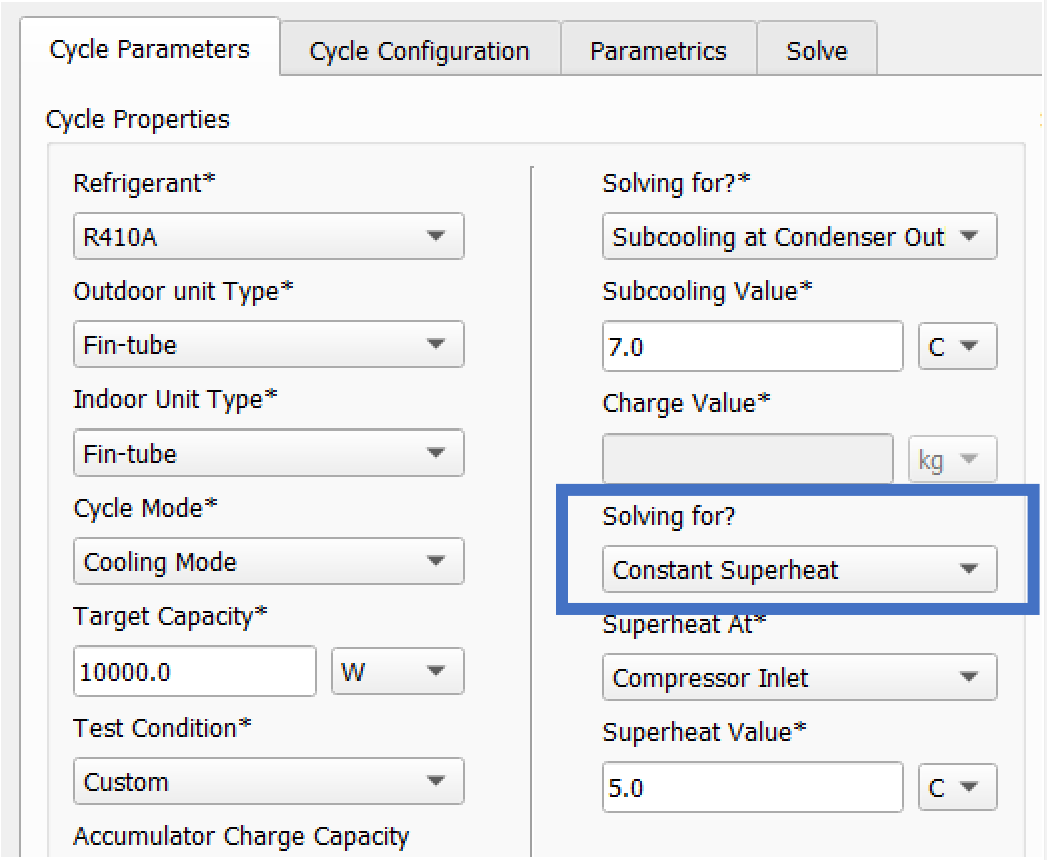
EGSim Solver Inputs

- This is the first input to the solver
- This can be:
  - Subcooling value at the condenser outlet
  - Cycle charge value (*currently disabled as void fraction models available in open literature are inaccurate and might produce unrealistic results*)



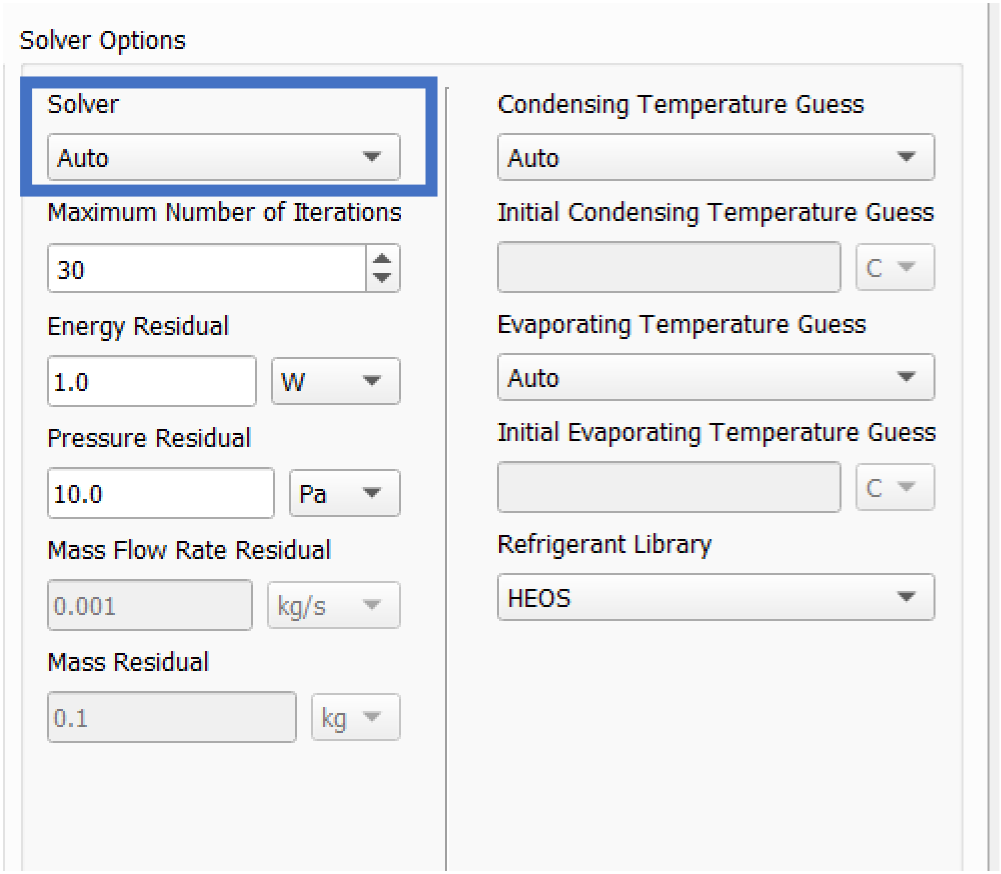
Expansion Device

- Currently the solver can model either a constant superheat (similar to a TXV) or a capillary tube
- If constant superheat is selected, a value should be defined either at the evaporator exit or the compressor inlet
- If the capillary tube is selected, no additional inputs would be required; however, the capillary tube design should be specified in the cycle Components (*this option is currently disabled as capillary tube models available in open literature are inaccurate and might produce unrealistic results*)



Solver

- This field is used to define the solver used for solving the cycle
- The “Auto” option will choose the most suitable solver according to current inputs



Solver: Options and Tradeoffs

Solver	Advantages	Disadvantages
Newton	Fast convergence	Suffers from overshooting and divergence while solving for saturation temperatures
Broyden 1	Very fast converge	Suffers from overshooting and divergence while solving for saturation temperatures

Solver: Options and Tradeoffs, cont.

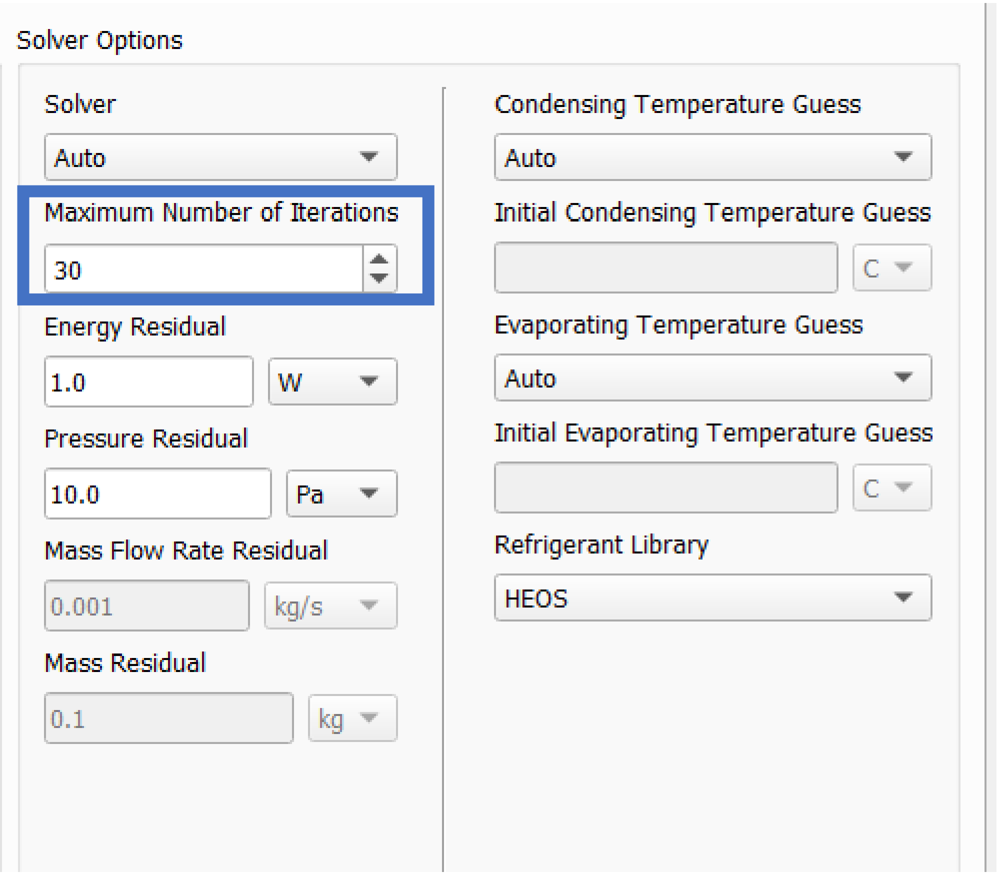
Solver	Advantages	Disadvantages
Broyden 2	Uses similar gradient evaluation to Broyden 1 with different implementation	Uses similar gradient evaluation to Broyden 1 with different implementation
Hybrid	Very stable solver; can be used for difficult cases where convergence could not be met with other solvers	Very slow as it solves the cycle 5 times before moving to the next inner iteration

Solver: Options and Tradeoffs, cont.

Solver	Advantages	Disdvantages
Least Squares	Stable with confined search domain to the possible temperatures	Very slow it solves the cycle 3 times before moving to the next inner iteration
Exciting Mixing	Fast convergence with some cases	Suffers from overshooting and divergence during solving for saturation temperatures

Maximum Number of Iterations

- This field is used to define the maximum number of iteration of the inner and outer solver loops
- Care should be taken when selecting the maximum number of iterations, a large number would provide an opportunity for the solution to converge, but would come at additional computational time
- This input is important when performing parametric studies since one or two cases of the parametric study might not be able to converge. Hence, using a large number will extend the calculation time



Energy Residual

- This field is used to define the allowable energy residual that will be used in the case of subcooling at condenser or constant superheat
- The cycle energy residual is calculated using the following equation

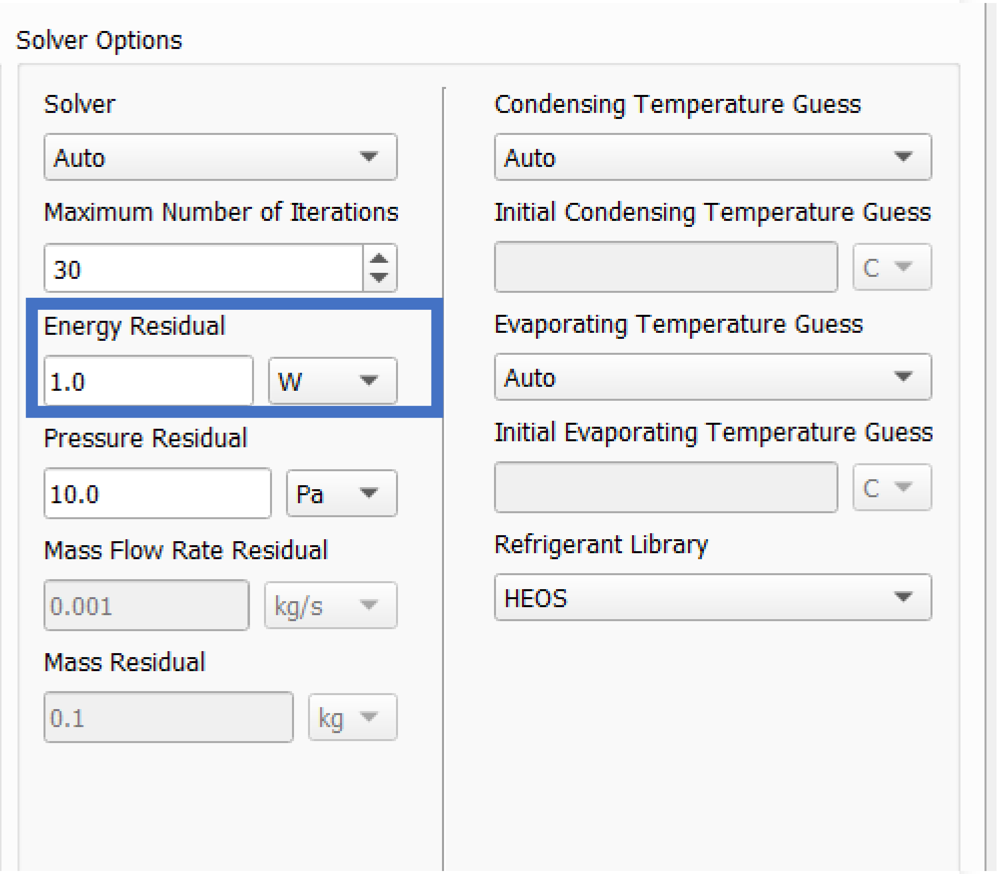
$$Residual = \left|\dot{m}_{ref} * \left(h_{actual} - h_{refrence}\right)\right|$$

Where:

$\dot{m}_{ref}$: *refrigerant mass flowrate*

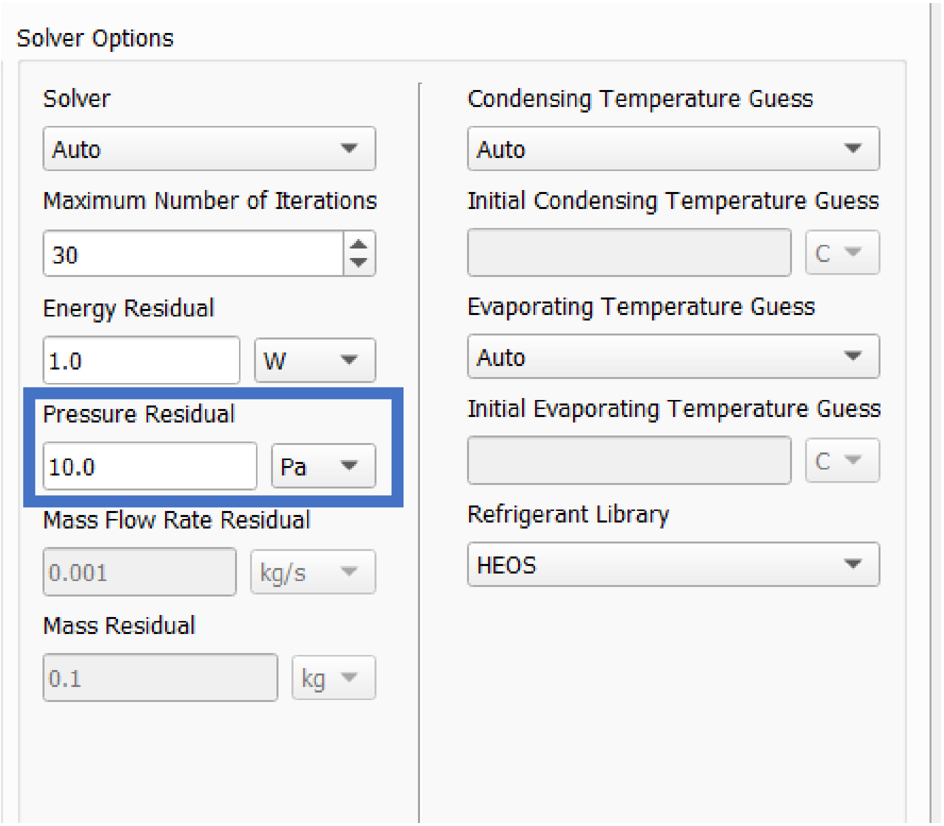
h_{actual} : *calculated enthalpy at the condenser exit, or evaporator exit, or compressor inlet*

$h_{refrence}$: *enthalpy at the condenser exit, or evaporator exit, or compressor inlet using the input condition*



Pressure Residual

- This field is used to define the allowable pressure residual that will be used for pressure drop
- The pressure drop residual is calculated using the following equation



$$Residual = \ |\Delta Pressure\ Drop_{new} - \Delta Pressure\ Drop_{old}\ |$$

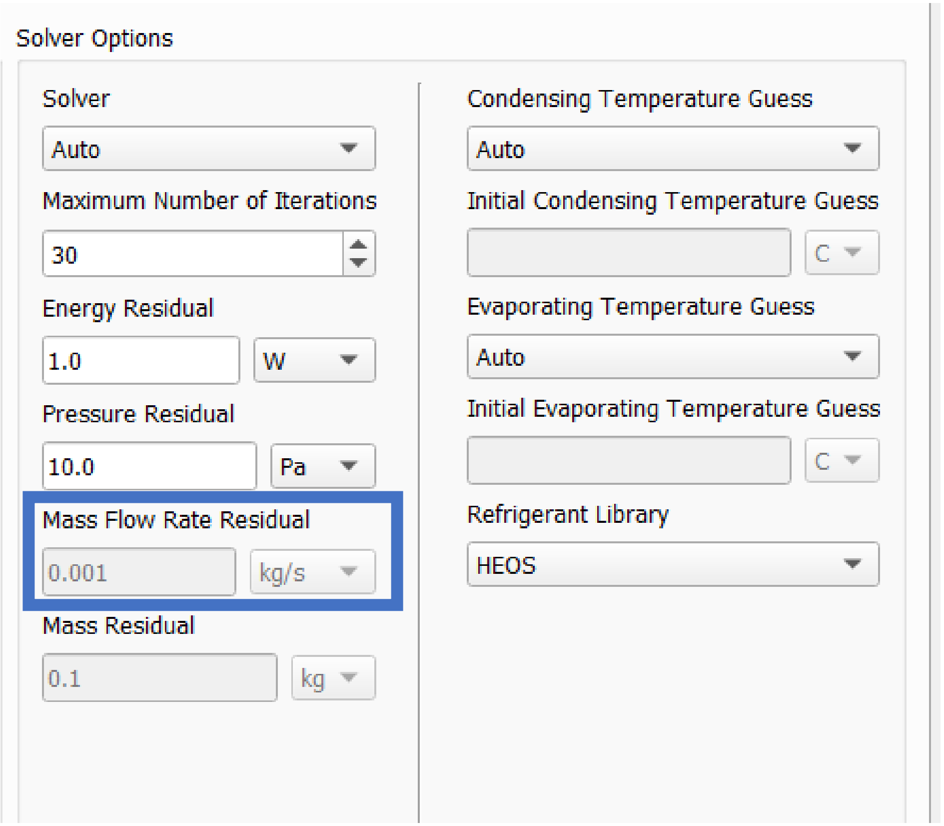
Where:

$\Delta Pressure\ Drop_{new}$: *total cycle pressure drop at iteration i+1 of outer loop*

$\Delta Pressure\ Drop_{old}$: *total cycle pressure drop at iteration i of outer loop*

Mass Flowrate Residual

- This field is used to define the allowable mass flowrate residual used in the case of capillary tube
- The mass flowrate residual is calculated using the following equation



$$Residual = \left|\dot{m}_{capillary} - \dot{m}_{compressor}\right|$$

Where:

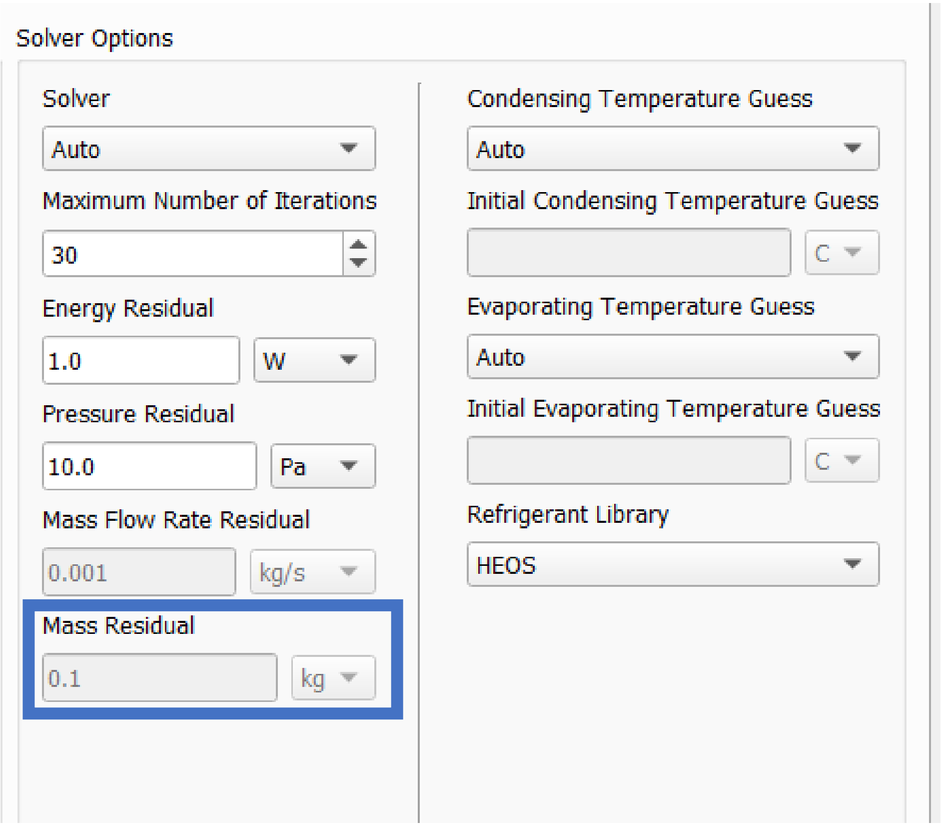
$\dot{m}_{capillary}$: *refrigerant mass flowrate calcualted from capillary tube model*

$\dot{m}_{compressor}$: *refrigerant mass flowrate calcualted from compressor model*

Mass Residual

- This field is used to define the allowable mass residual used in the case of system charge
- The mass residual is calculated using the following equation

$$Residual = \left| m_{reference} - m_{calculated} \right|$$

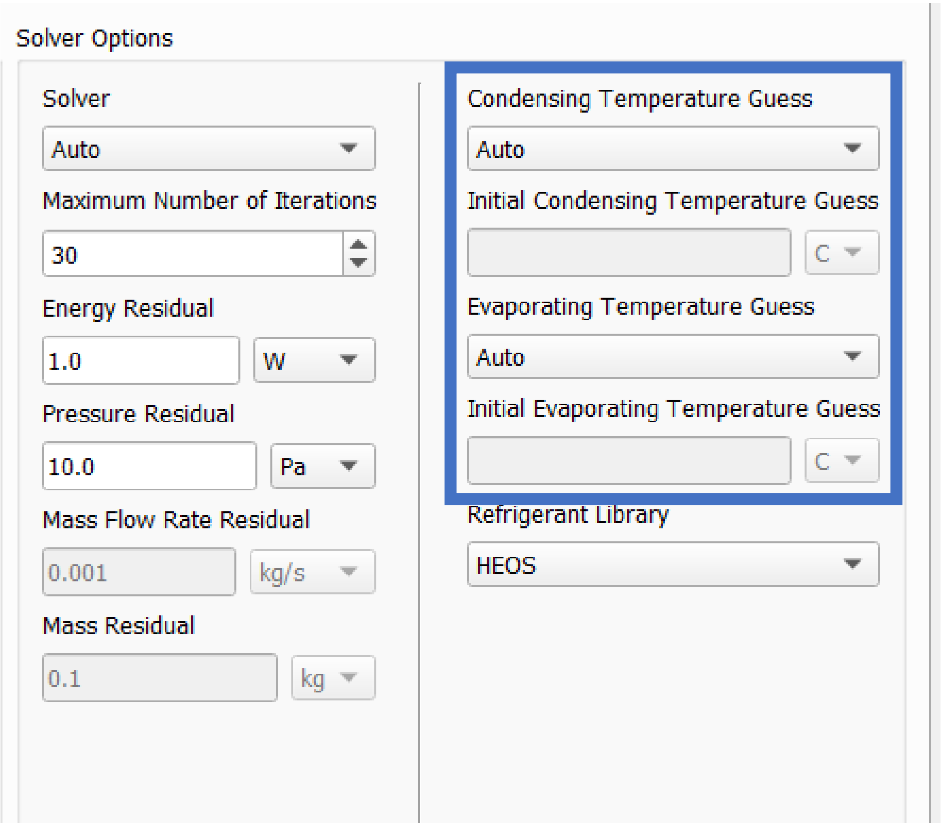


Where:

$m_{reference}$: *refrigerant system charge defined by the user*

$m_{calculated}$: *refrigerant system charge calculated for the actual cycle*

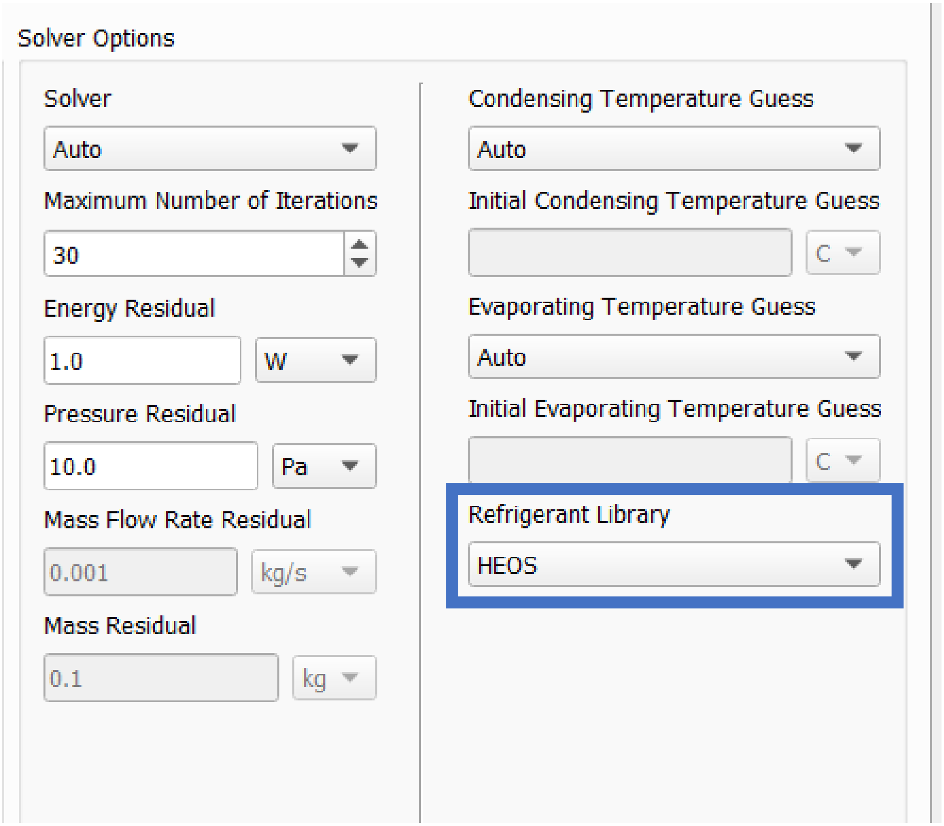
Condensing and Evaporating Temperature Initial Guess Values

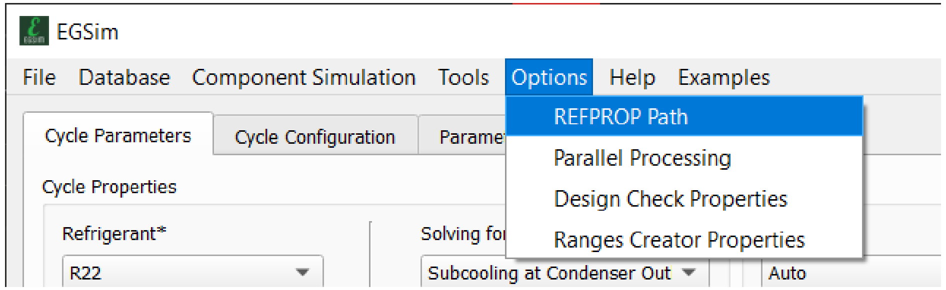


- These fileds define how the initial guess for the condensing and evaporating temperatures will be handled
- If Auto is chosen, then EGSim will try to guess the suitable initial values depending on the defined heat exchangers and the compressor model
- If manual is chosen, input values will be used

Refrigerant library

- This field defines how refrigerant properties will be calculated
  - “HEOS” which stands for Helmholtz equations of state is included with EGSim and can be used for free
  - “REFPROP” [1] which is the industry-standard reference for refrigerant properties can be used if a licensed version is available on the computer
    - “REFPROP” path can be defined from
      - Menu bar => Options => REFPROP Path
    - “REFPROP” is considered more accurate





References

- [1] Lemmon, E.W., Bell, I.H., Huber, M.L., McLinden, M.O. NIST Standard Reference Database 23: Reference Fluid Thermodynamic and Transport Properties-REFPROP, Version 10.0, National Institute of Standards and Technology, Standard Reference Data Program, Gaithersburg, 2018.

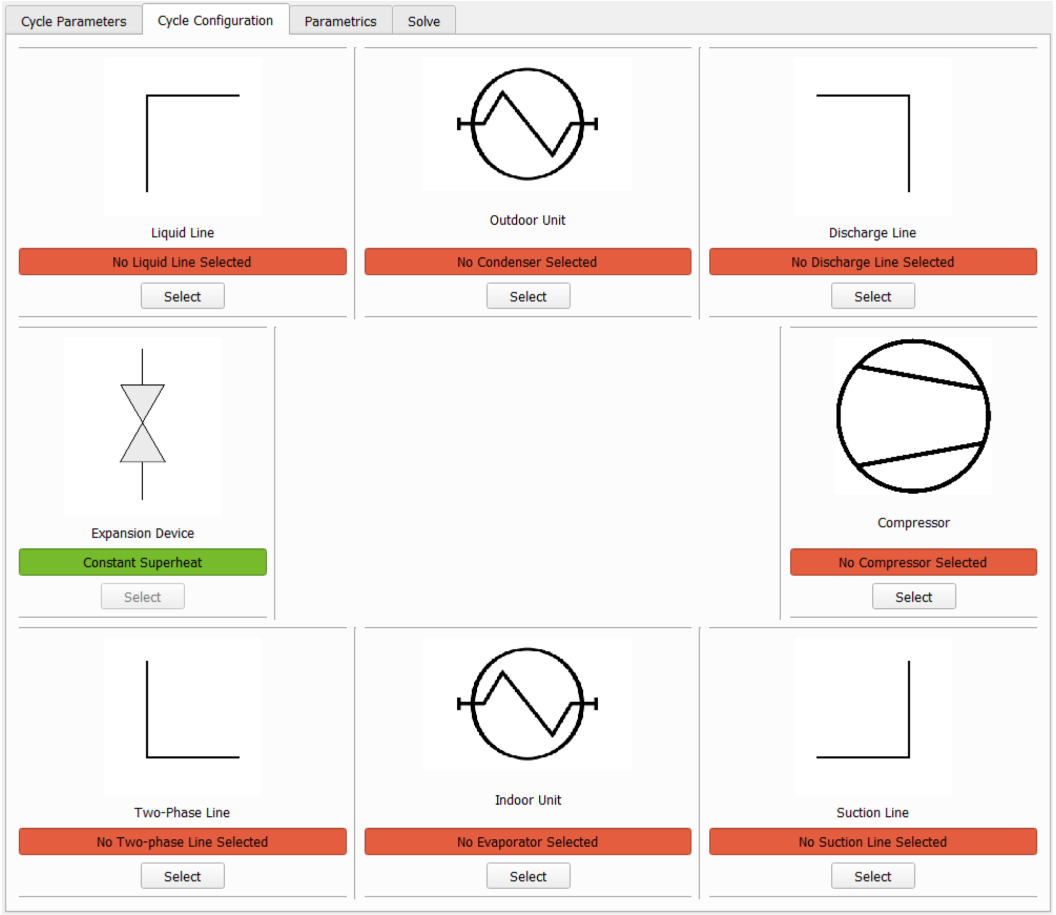
  https://www.nist.gov/srd/refprop

3. Cycle Components

EGSim User Guide

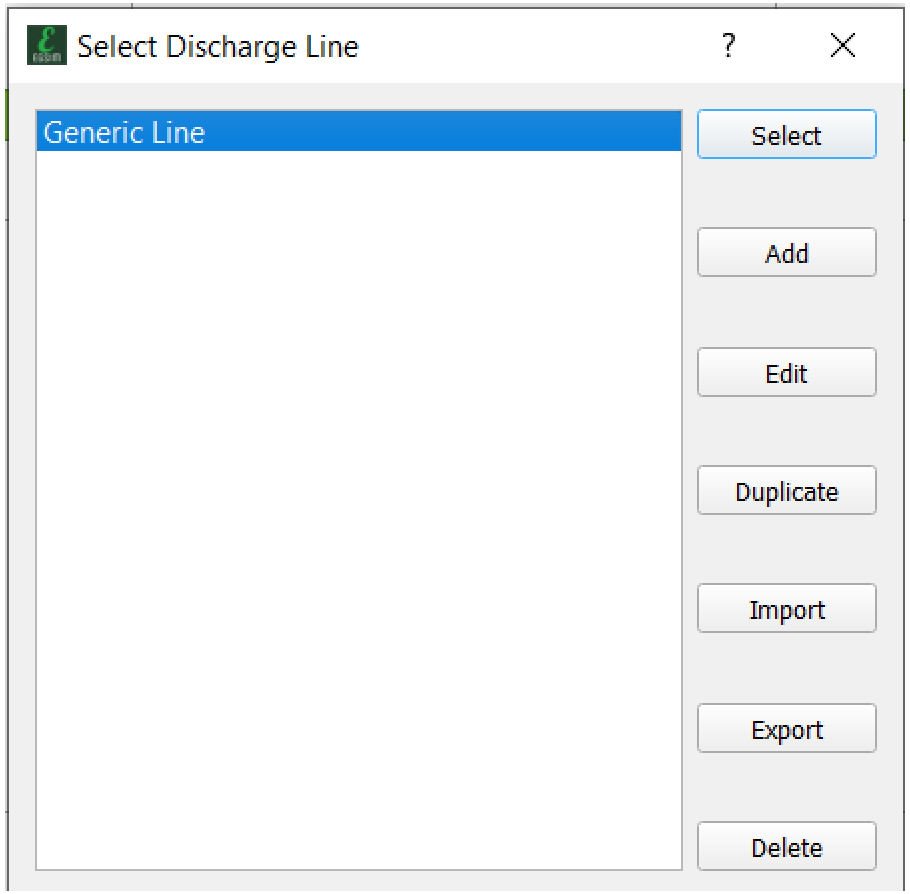
Overview

- This tab can be used to define the cycle components.
- On each component, the user can select, edit, add any of the following components:
  - Indoor unit
  - Outdoor unit
  - Compressor
  - Connecting lines
  - Capillary (if used)



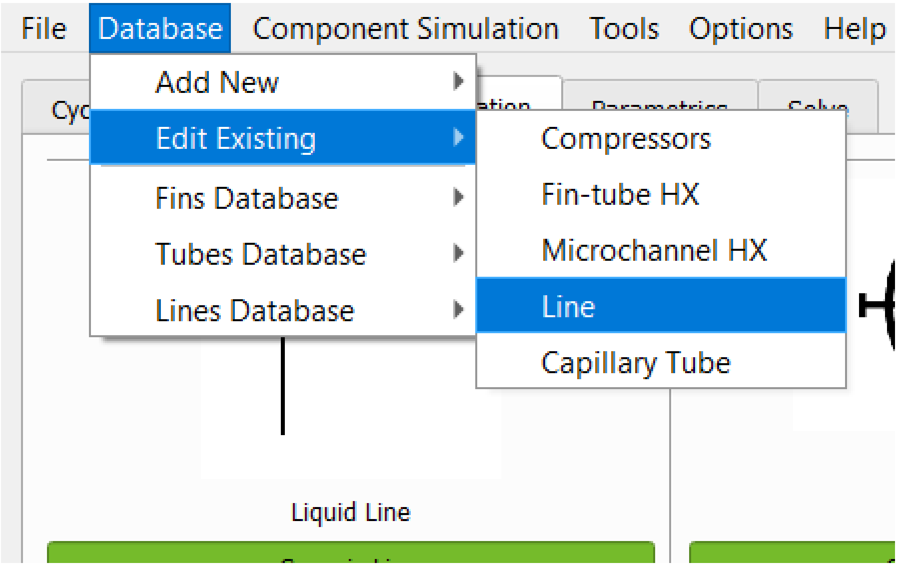
Select Window

- If select button of any of the components is clicked, a window showing the available predefined components of this type available with several button to control the chosen components:
  - Select: selects this component
  - Add: creates a new component
  - Edit: edits selected component
  - Duplicate: creates a duplicate copy of selected component
  - Import: imports component from external file
  - Export: exports selected component to external file
  - Delete: deletes selected component from database



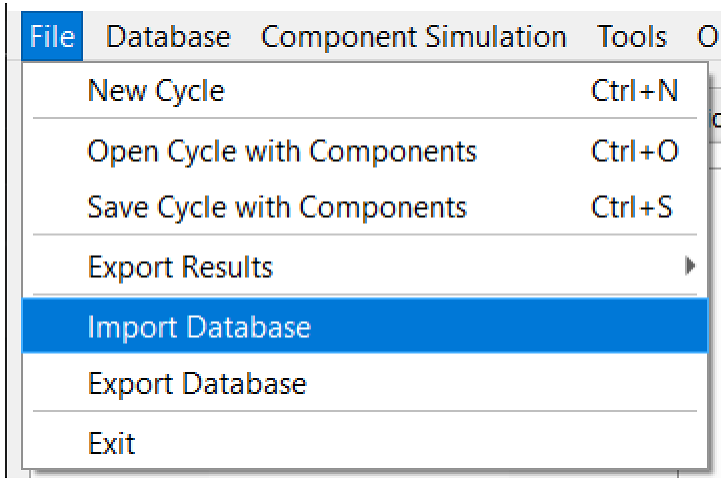
Select Window, cont.

- A similar window for component control in the database can be triggered from:
  - Database => Edit existing



Components' Database Handling

- You can export current complete database using:
  - File => Export Database
- You can also import a complete database using:
  - File => Import Database
- <u>Note: Importing a database will delete all current components in the database</u>



3.1 Compressor

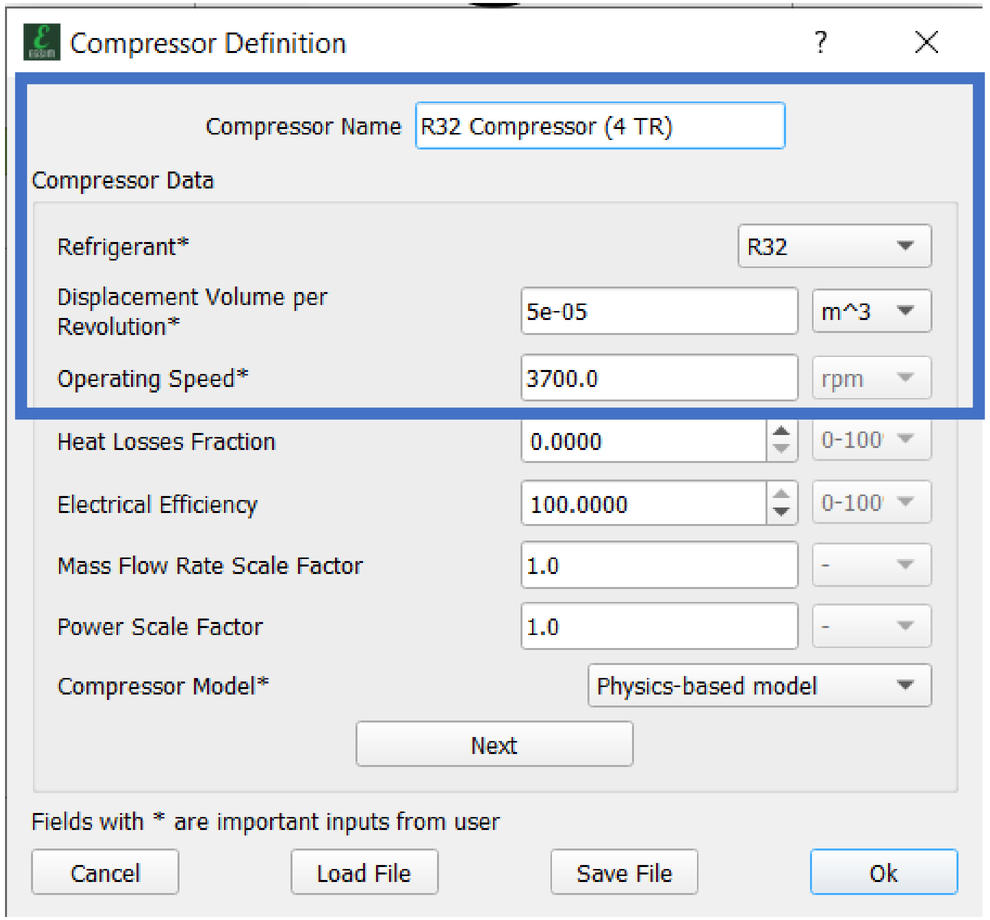
EGSim User Guide

Overview

- The compressor is considered the heart of any vapor compression cycle. It is one of the main drivers for the cycle performance and pumps the refrigerant throughout the system
- An accurate definition of the compressor will yield an accurate cycle solution
- The following compressor models are available with EGSim:
  - Physics-based Model
  - AHRI 10-coefficiencts model
  - Compressor Map model
- These models are used to evaluate the refrigerant mass flowrate and compressor power

Compressor Inputs

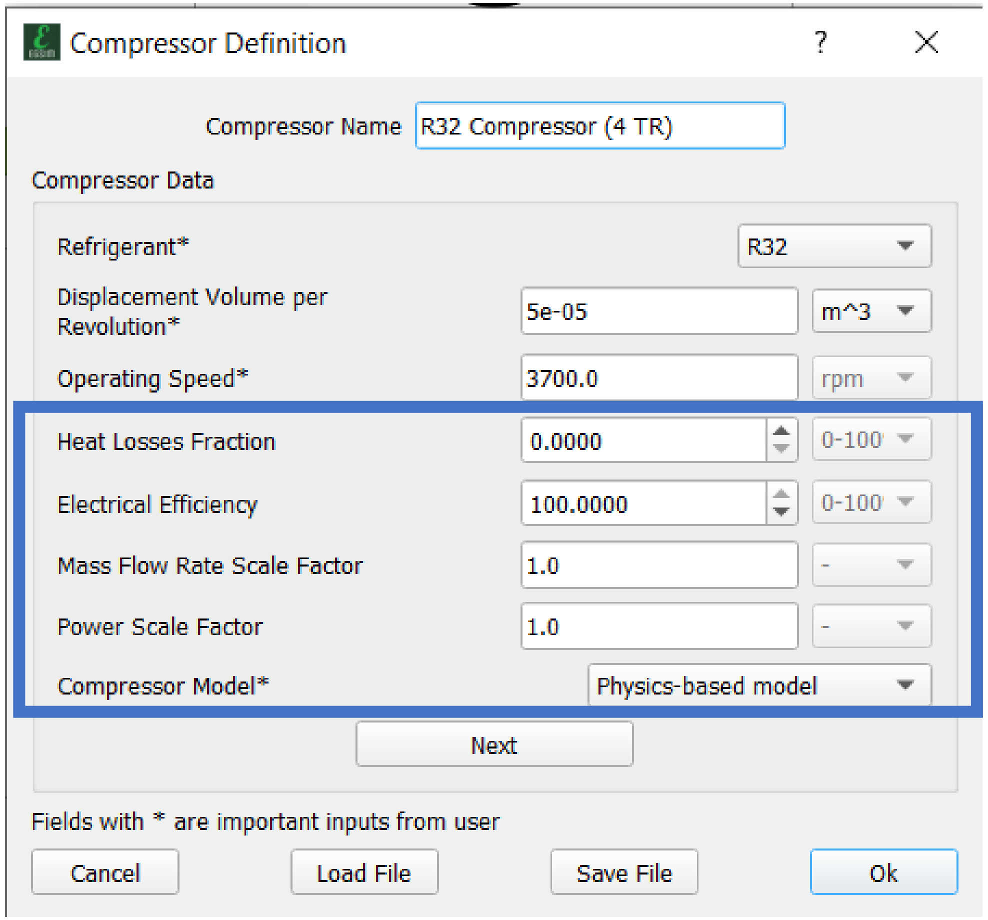
- **Compressor Name:** Name used in the database
- **Refrigerant:** refrigerant to be used with the compressor; choose "Not specified" in case no specific refrigerant is to be chosen
- **Displacement Volume per revolution:** the swept volume of the compressor per revolution
- **Operating Speed:** the running speed of the compressor



Compressor Inputs, cont.

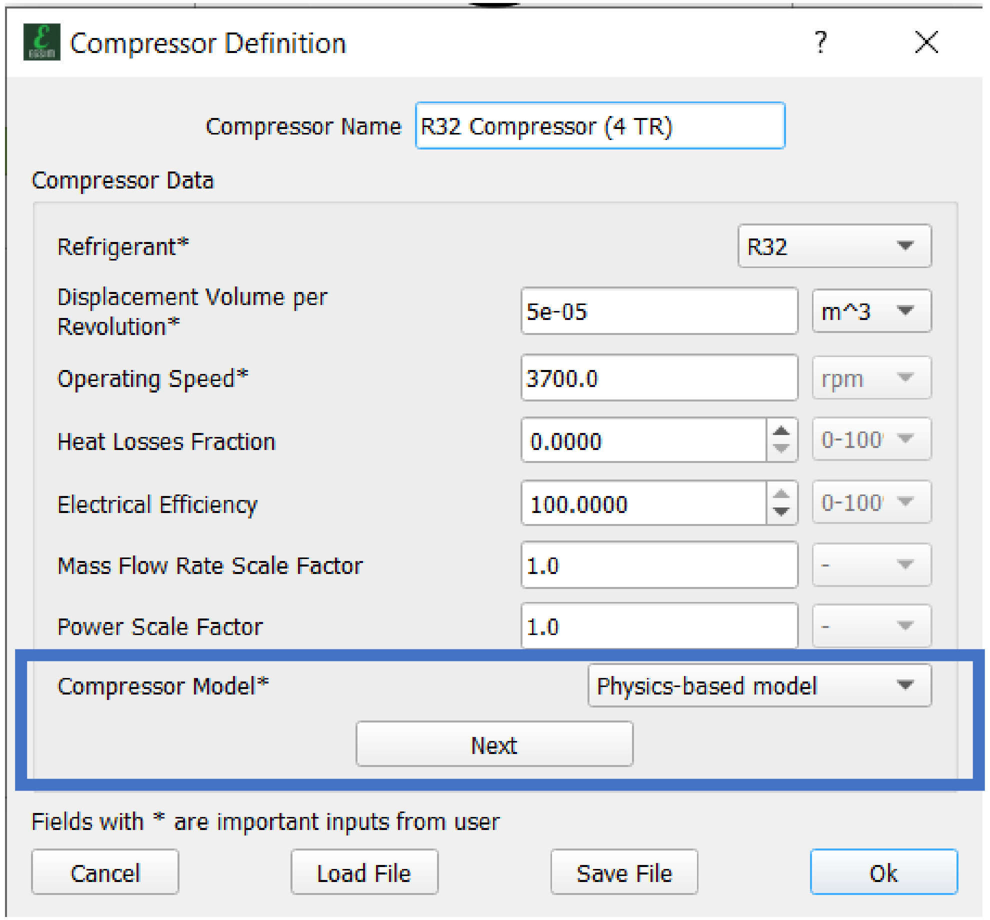
- **Heat Losses fraction:** fraction of compressor mechanical power that will be dissipated as heat
- **Electrical Efficiency:** efficiency of the compressor motor/driver (e.g. VFD efficiency)
- **Mass flowrate scale factor*:** a multiplier of the refrigerant mass flowrate
- **Power flowrate scale factor*:** a multiplier of the compressor power

* These could be used to simulate compressors in parallel or to calibrate the system performance for validation purposes



Compressor Inputs, cont.

- **Compressor model:** the user can select one of three available compressor models
  - Physics-based Model
  - AHRI 10-coefficients model
  - Compressor map model
- when the user presses “Next”, he/she will be guided to define additional inputs based on the selected compressor model

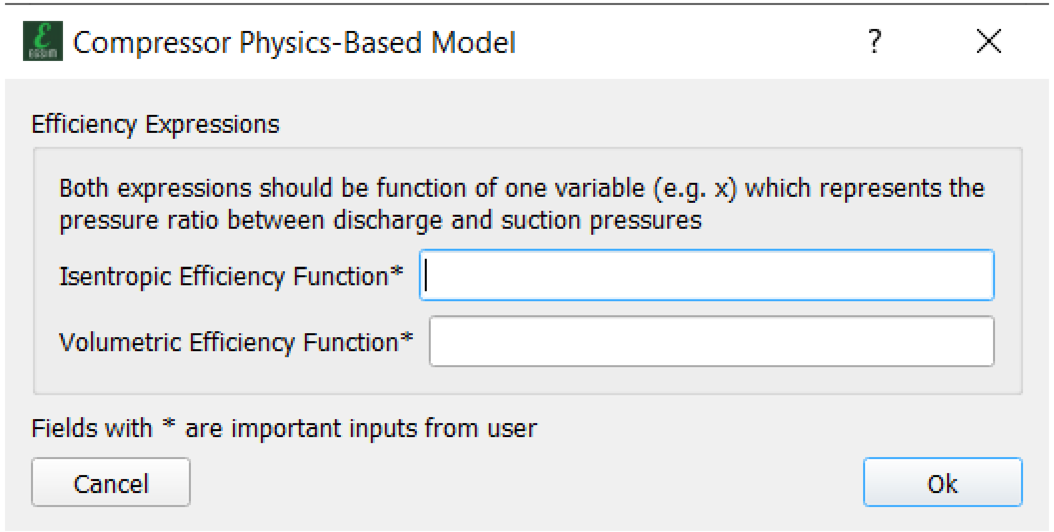


3.1.1 Physics-Based Model

EGSim User Guide

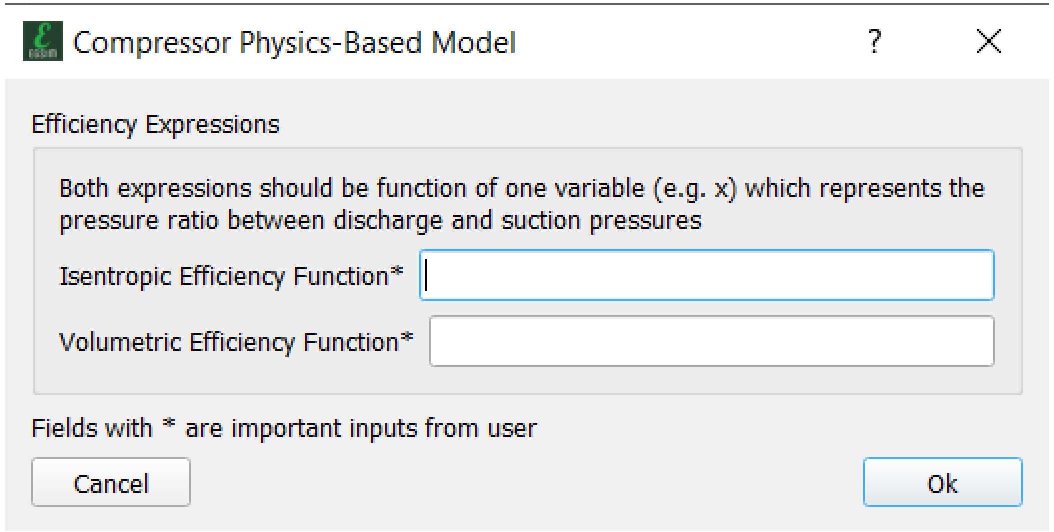
Physics-Based Model

- Physics-based model is used to define a compressor using:
  - Isentropic efficiency used in calculating compressor power
  - Volumetric efficiency used in calculating refrigerant mass flowrate
- Both efficiencies can be defined as any mathematical expression of the compressor pressure ratio



Physics-Based Model, cont.

- The resulting value of the expression should give efficiency in the range of 0 to 1, *the solver will raise an error if the efficiency is outside this range*
- For volumetric efficiency, the value might exceed 1, *the solver will raise an error if the volumetric efficiency is less than 0 or greater than 1.5*
- Example values are:
  - 0.6
  - -0.11*x^2 + 0.2 * x + 0.7



3.1.2 AHRI 10-Coefficients Model

EGSim User Guide

AHRI 10-Coefficients Model

- The AHRI 10-coefficients model is used to define compressor using:
  - 10-coefficients for refrigerant mass flow rate
  - 10-coefficients for compressor power
- The 10-coefficients are based on a third order nonlinear multivariate model of refrigerant mass flow rate and power

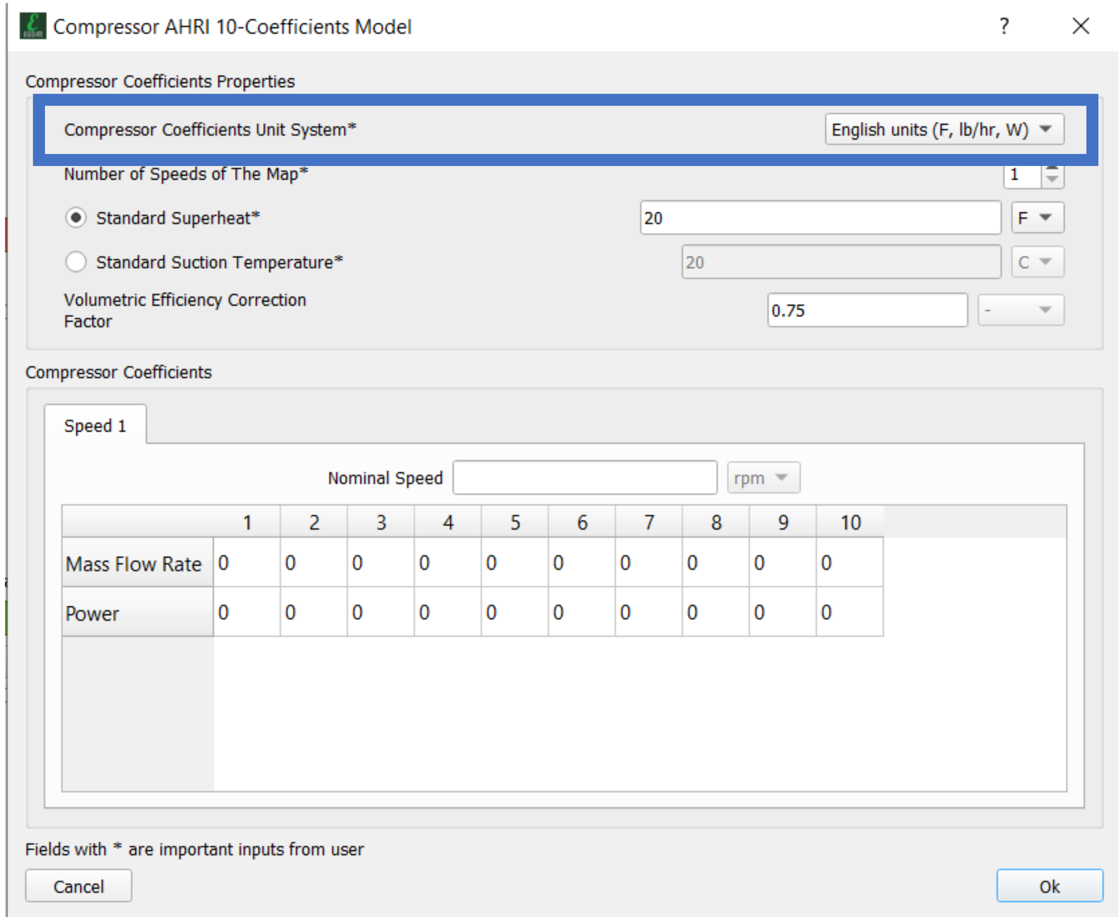
$$Z = C1 + C2 * x + C3 * y + C4 * x\char`^2 + C5 * x * y + C6 * y\char`^2 + C7 * x\char`^3 + C8 * x\char`^2 * y + C9 * x * y\char`^2 + C10 * y\char`^3$$

z= refrigerant mass flow rate or compressor power, x = Evaporating temperature (˚C or ˚F), y = Condensing temperature (˚C or ˚F)

AHRI 10-Coefficients Model, cont.

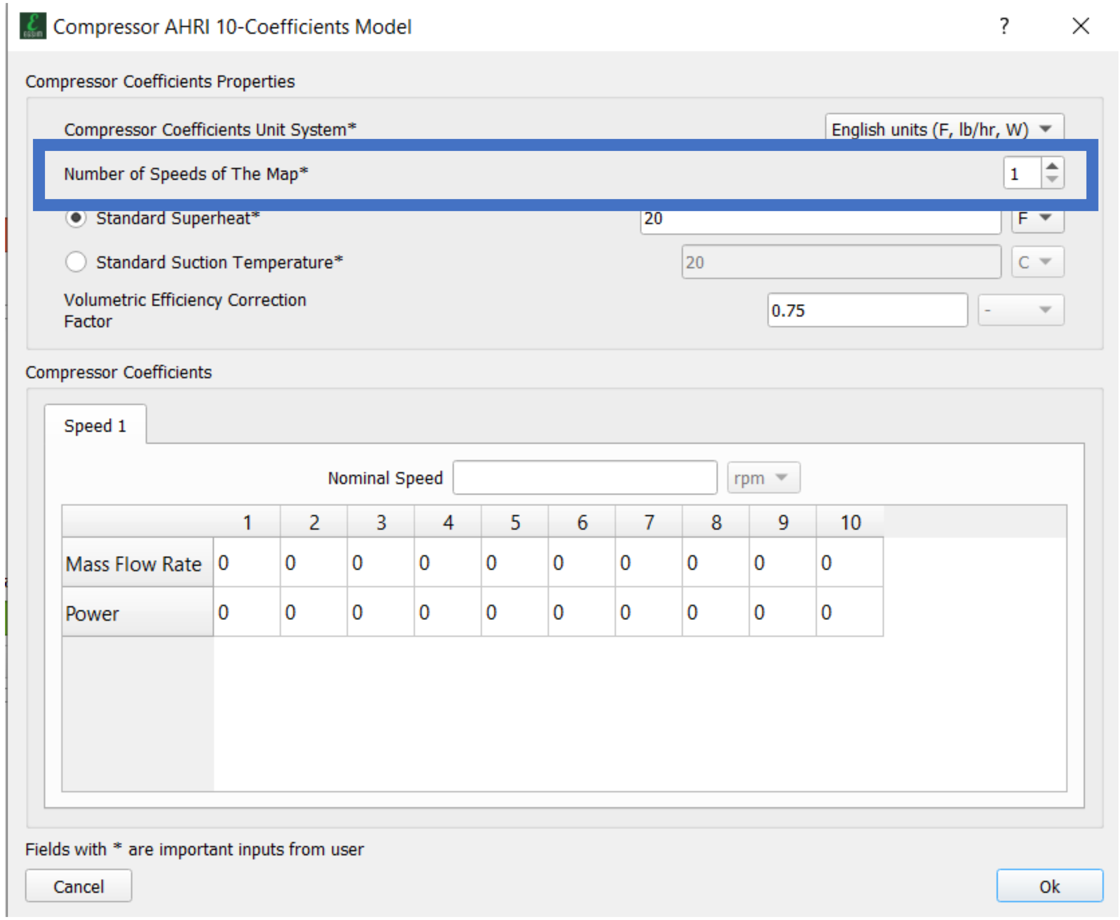
- First, you have to define the coefficients unit system (temperature, mass flow rate, power)

Option #	Temperature	Mass flow rate	Power
1. English units	˚F	lb/hr	W
2. SI units	˚C	kg/s	W
3. SI units	˚C	kg/hr	W



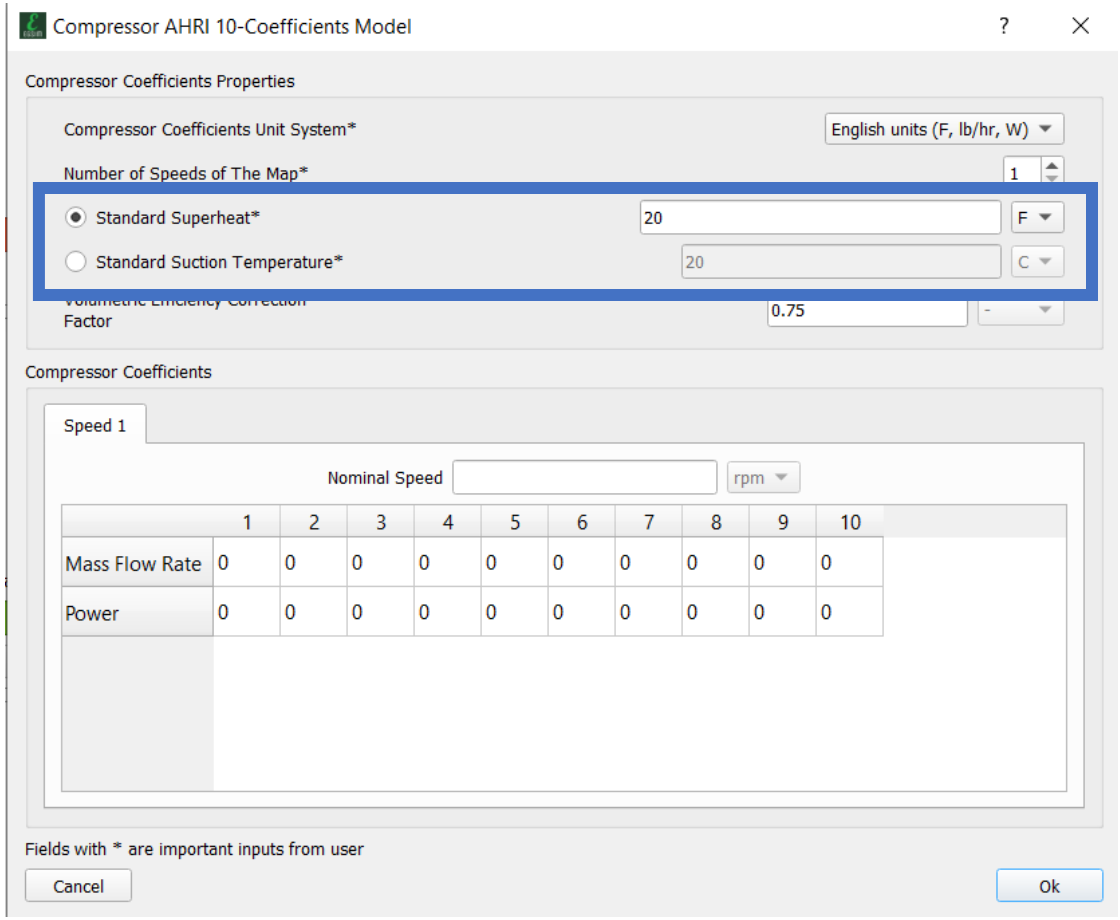
AHRI 10-Coefficients Model, cont.

- Number of speeds of the map defines the number of compressor speeds used to build the compressor coefficient
- For each speed, 10-coefficients for mass flow rate and power must be entered
- For single speed compressor, the number of speeds should be 1
- For variable speed compressor, to be able to simulate the compressor at different speeds, the map should cover those speeds in its range



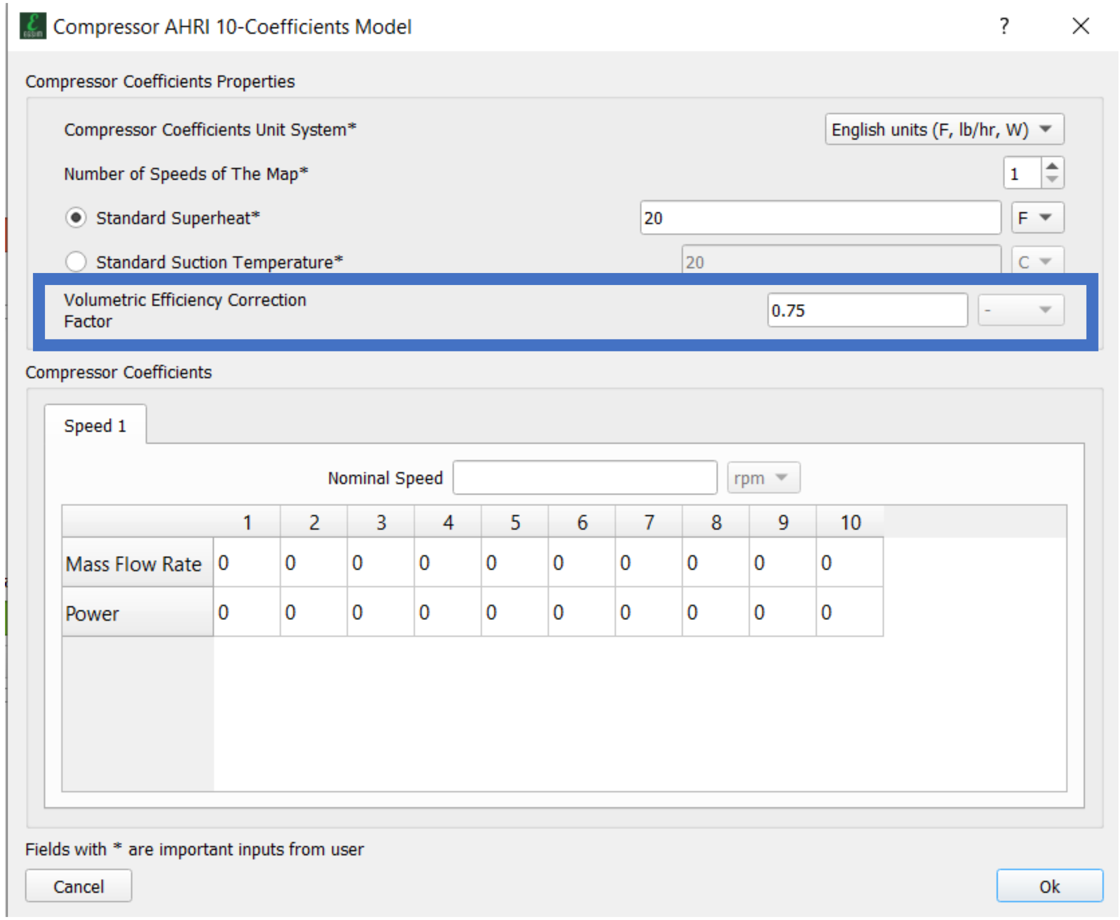
AHRI 10-Coefficients Model, cont.

- You have to define either standard superheat or standard suction temperature (return gas temperature) based on the data used to generate the coefficients
- This value will be considered for the superheat correction for refrigerant mass flow rate and the compressor power



AHRI 10-Coefficients Model, cont.

- The volumetric efficiency correction factor depends on the volumetric efficiency and is typically designated by the compressor supplier
- A value of 0.75 is usually used



AHRI 10-Coefficients Model, cont.

- Mass flow rate correction

$$\dot{m}_{corrected} = \left(1 + F_v\left[\frac{v_{standard}}{v_{actual}} - 1\right]\right) * \dot{m}_{rated}$$

Where:

$\dot{m}_{corrected}$: *corrected refrigerant mass flow rate*

F_v: *volumetric efficiency correction factor*

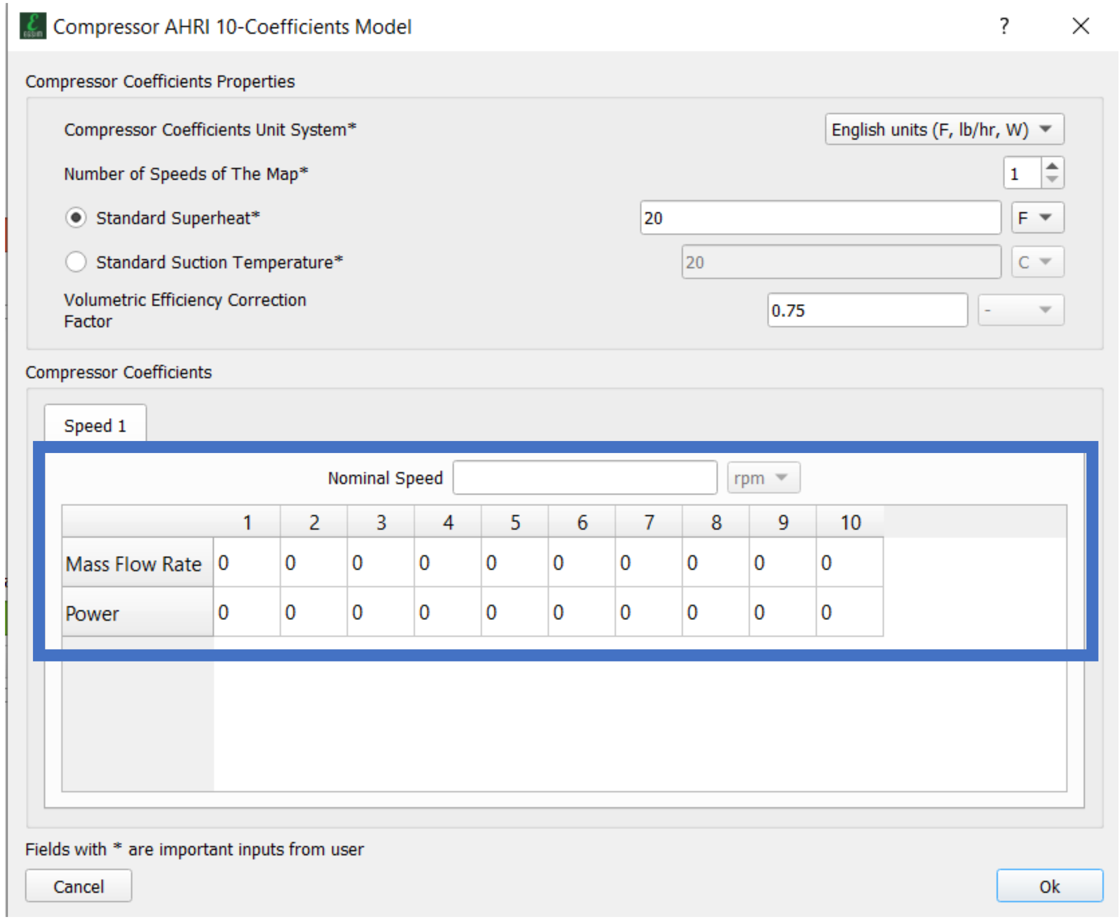
$v_{standard}$: *suction specific volume at standard condition*

v_{actual}: *suction specific volume at actual condition*

$\dot{m}_{rated}$: *rated refrigerant mass flow rate from the compressor map*

AHRI 10-Coefficients Model, cont.

- For each speed, you will have to enter the nominal speed value, and the 10 coefficients for refrigerant mass flow rate and compressor power
- If single speed compressor is used, enter the operating speed of the compressor, which should match the operating speed chosen before at the compressor definition main page (section 3.)
- <u>You should make sure that the compressor map covers your simulation range of evaporating and condensing temperatures</u>



3.1.3 Compressor Map Model

EGSim User Guide

Compressor Map Model

- The compressor map model uses the compressor performance at different evaporating and condensing temperatures to produce the AHRI 10-coefficients
- You will need to have two CSV files for refrigerant mass flow rate and compressor power
- Each file will consist of 3 columns:
    - Column 1: condensing temperature
    - Column 2: evaporating temperature
    - Column 3: value of mass flow rate or power

	A	B	C
1	Condensing Temperature	Evaporating Temperature	Value
2	45	5	20
3	45	10	30
4	50	5	40
5	50	10	50
6	..	..	..
7	..	..	..
8			

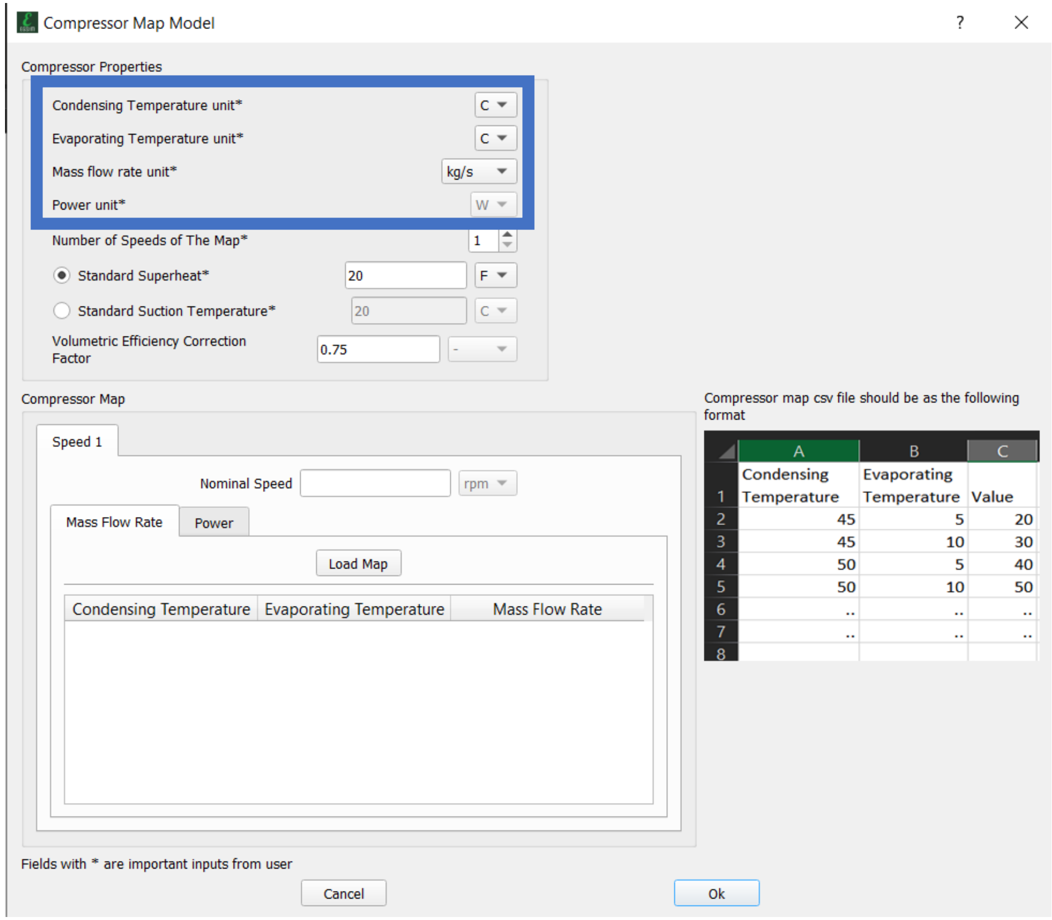
Compressor Map Model, cont.

- Excel or other software can be used to produce the csv files
- <u>Note that EGSim will skip the first row as this is typically reserved for table heading</u>
- <u>You should make sure that the compressor map covers your simulation range of evaporating and condensing temperatures</u>

	A	B	C
1	Condensing Temperature	Evaporating Temperature	Value
2	45	5	20
3	45	10	30
4	50	5	40
5	50	10	50
6	..	..	..
7	..	..	..
8			

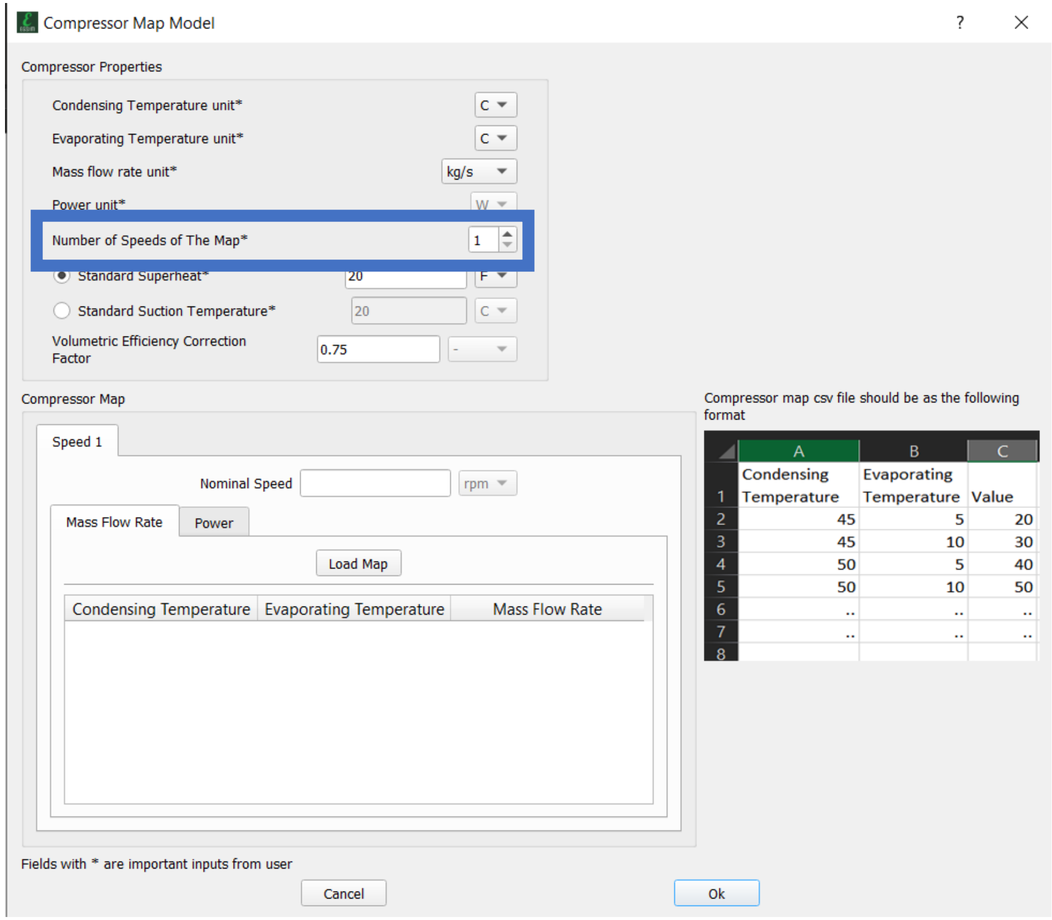
Compressor Map Model, cont.

- **Condensing and Evaporating temperatures units:** the units of the temperatures used in the csv files
- **Mass flow rate unit:** the unit of mass flow rate in the csv file
- **Power unit:** only W is available



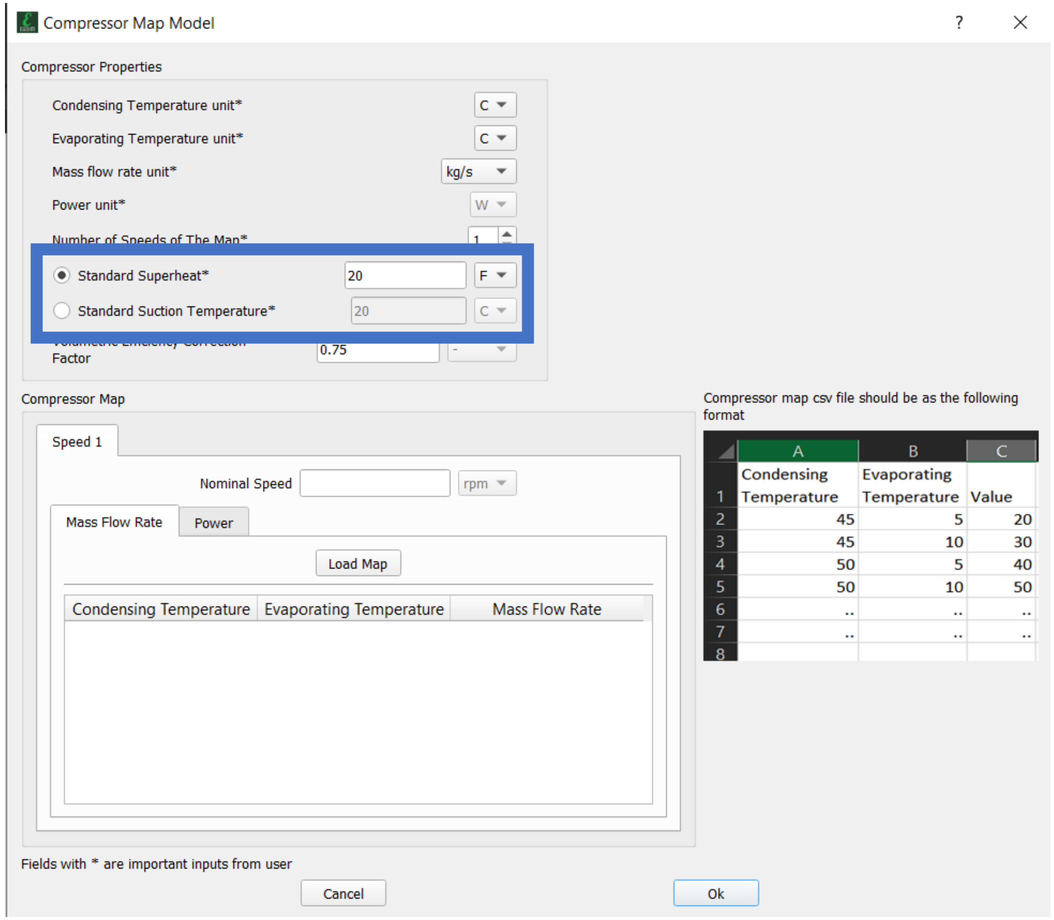
Compressor Map Model, cont.

- **Number of speeds of the map:** in case of variable speed compressor, you will have to define the compressor at different speeds covering the simulation range
- For each speed, you will need the two CSV files for mass flow rate and power
- For single speed compressor, choose number of speeds of the map to be 1



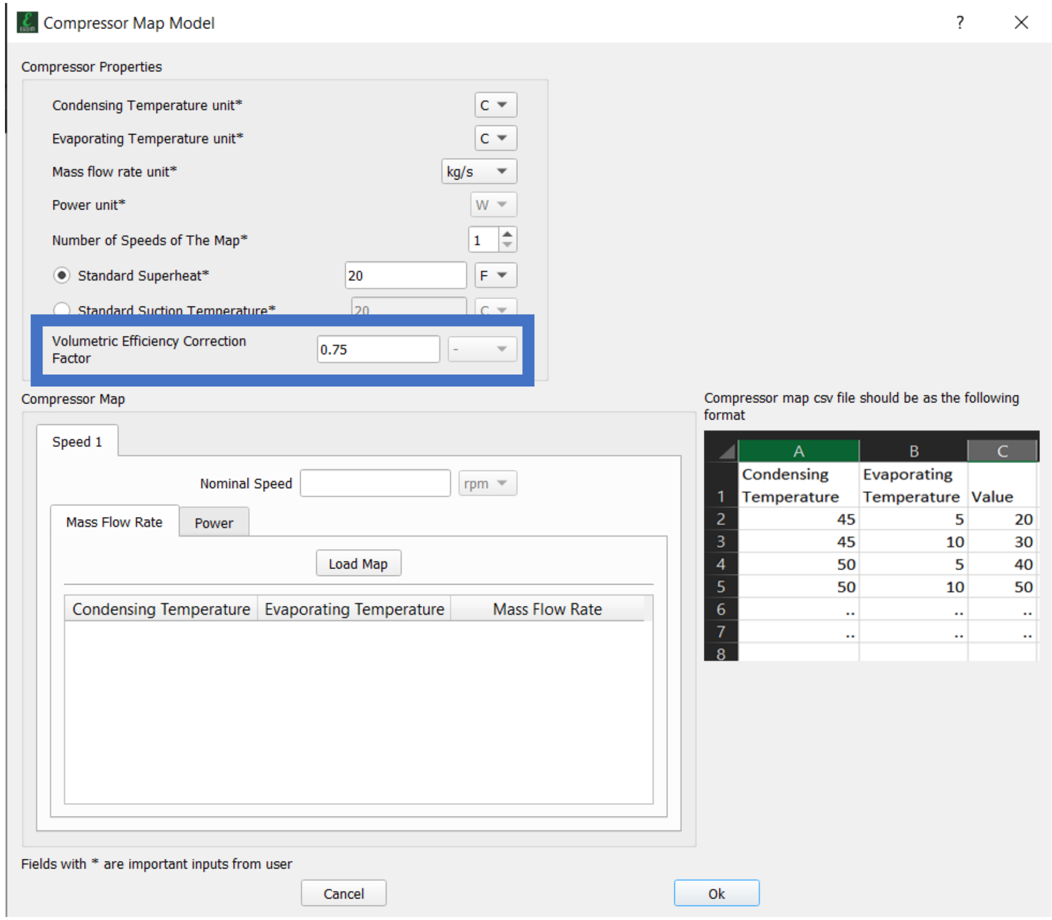
Compressor Map Model, cont.

- You have to define either standard superheat or standard suction temperature (return gas temperature) based on the data used to generate the coefficients
- This value will be considered for the superheat correction for refrigerant mass flow rate and the compressor power



Compressor Map Model, cont.

- The volumetric efficiency correction factor depends on the volumetric efficiency and is typically designated by the compressor supplier
- A value of 0.75 is usually used



Compressor Map Model, cont.

- Mass flow rate correction

$$\dot{m}_{corrected} = \left(1 + F_v\left[\frac{v_{standard}}{v_{actual}} - 1\right]\right) * \dot{m}_{rated}$$

Where:

$\dot{m}_{corrected}: corrected\ refrigerant\ mass\ flow\ rate$

$F_v: volumetric\ efficiency\ correction\ factor$

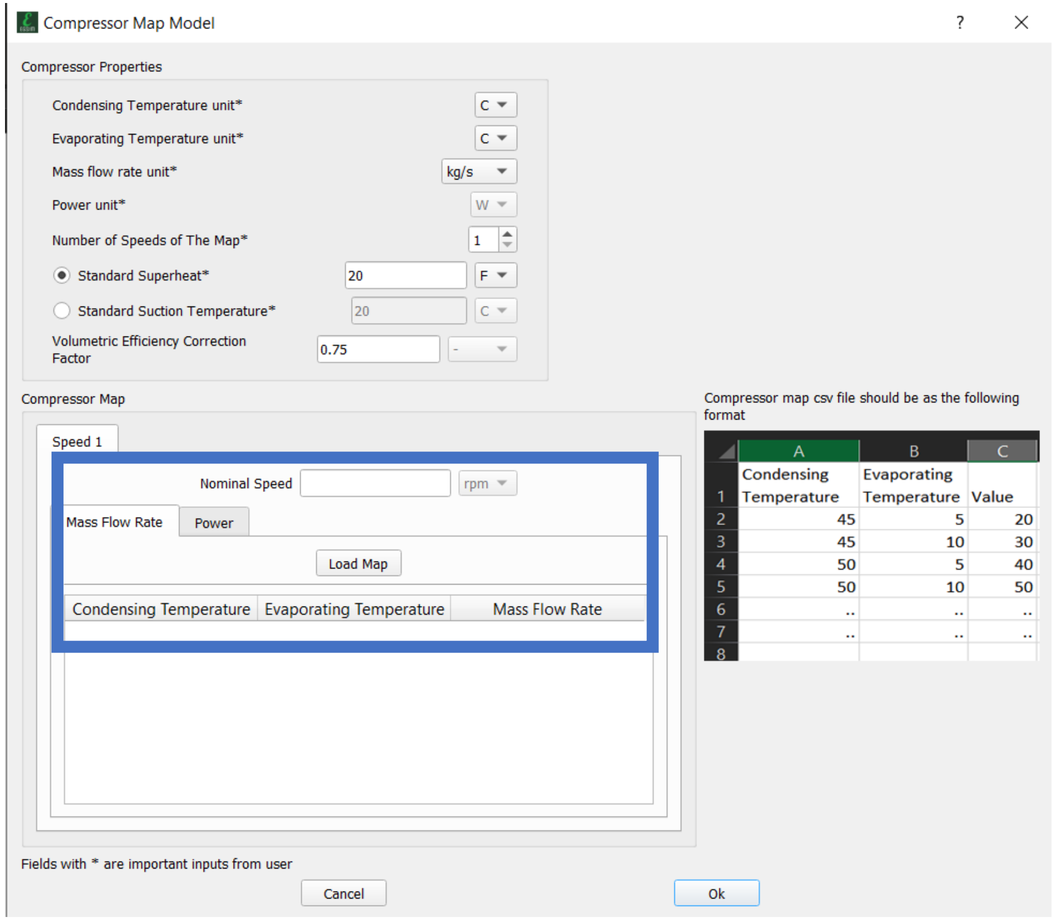
$v_{standard}: suction\ specific\ volume\ at\ standard\ condition$

$v_{actual}: suction\ specific\ volume\ at\ actual\ condition$

$\dot{m}_{rated}: rated\ refrigerant\ mass\ flow\ rate\ from\ the\ compressor\ map$

Compressor Map Model, cont.

- For each speed, you will have to import a file for the mass flow rate and a file for power
- You will have the ability to see the data that was imported to make sure it was imported correctly
- Note that EGSim will skip the first row as this is the table heading
- If single speed compressor is used, enter the operating speed of the compressor, which should match the operating speed chosen before at the compressor definition main page (section 3.)

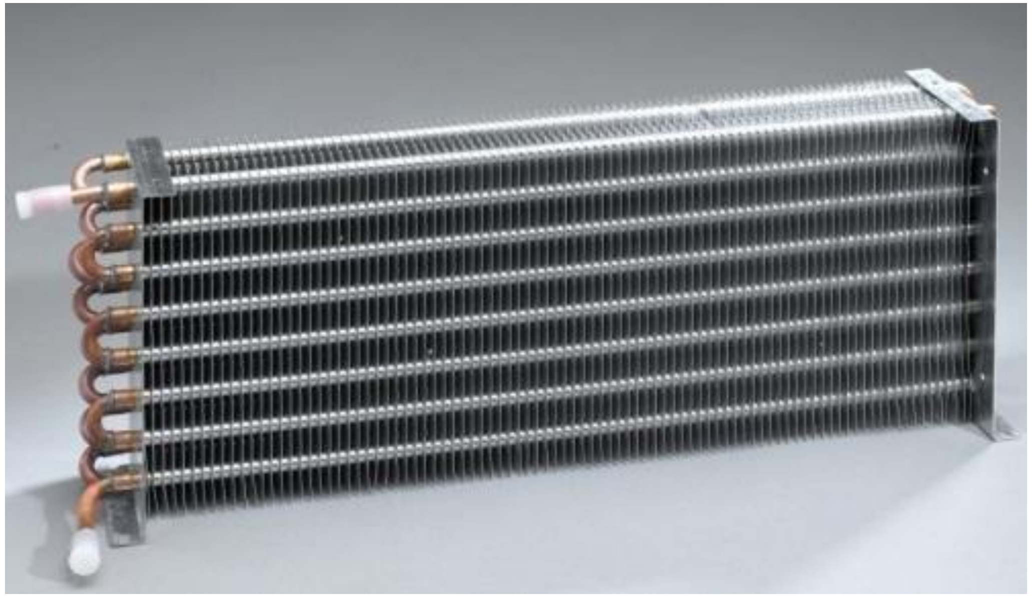


3.2 Fin-Tube Heat Exchangers

EGSim User Guide

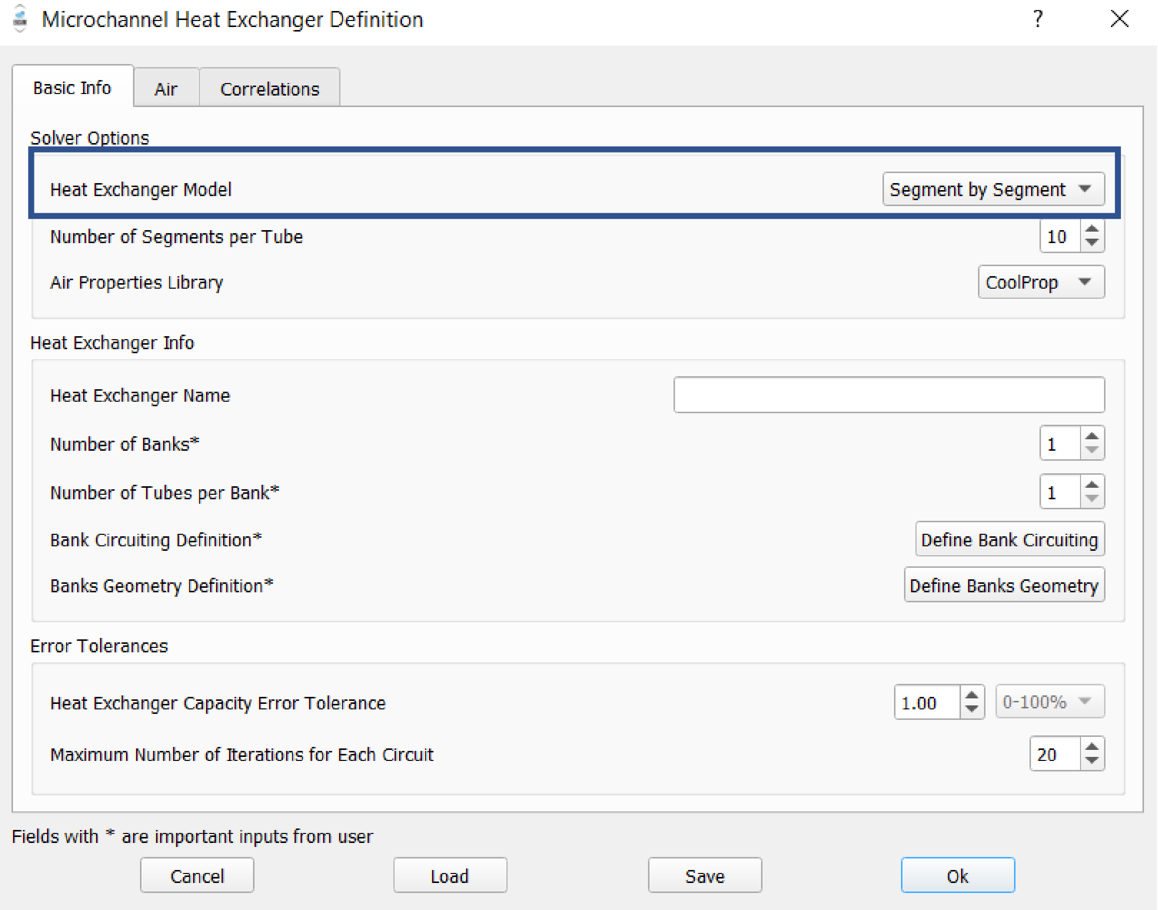
Overview

- Fin-tube heat exchangers is one of the two options available for heat exchangers models available in EGSim
- Fin-tube heat exchangers can be used for either the condenser or the evaporator
- In fin-tube heat exchangers, we define the detailed geometry and air flowrate inlet conditions



Heat Exchanger Model

- The **Heat Exchanger Model** allows the user to define solver options
  - Phase-by-phase solver
  - Segment-by-segment solver

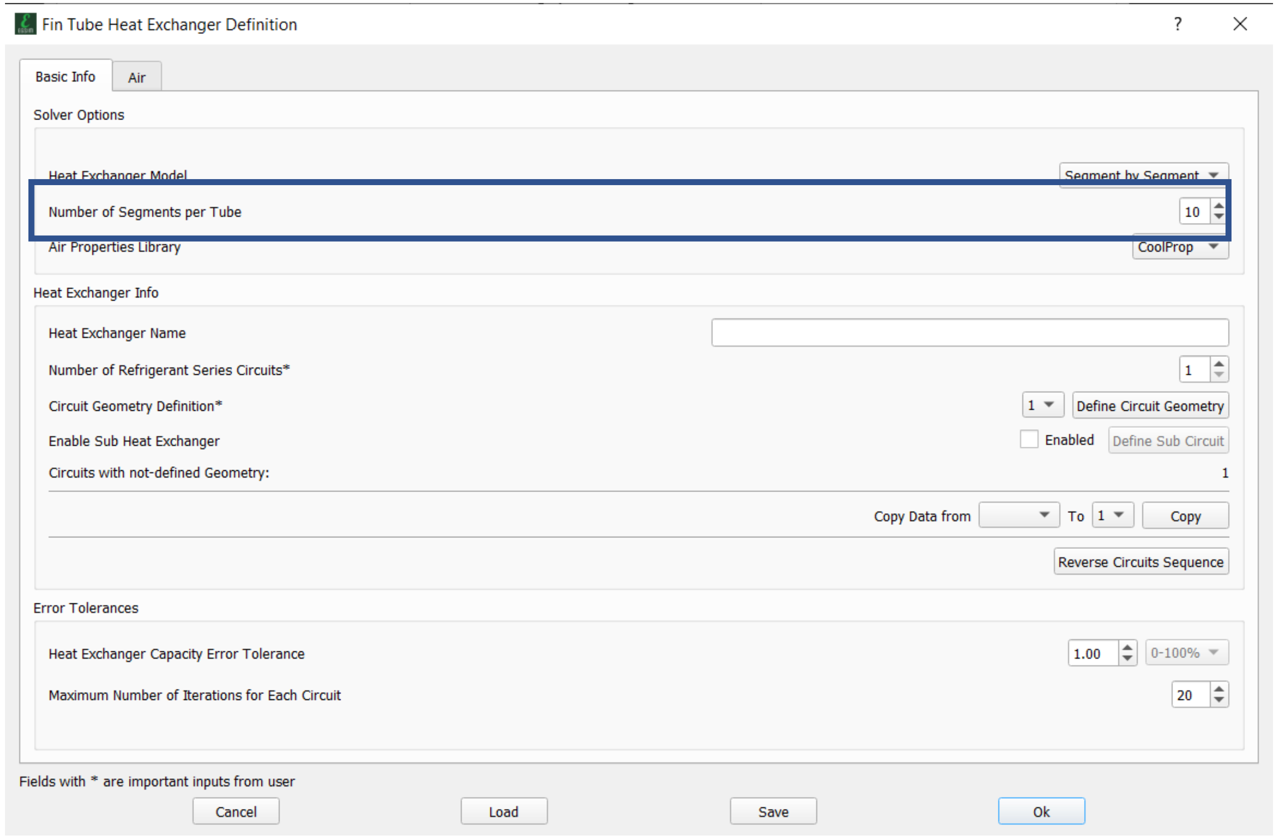


Phase-by-Phase Solver

- The heat exchanger is divided into a number of segments corresponding to the number of refrigerant phases
- The maximum number of segments is 3: superheated vapor, 2-phase, and subcooled liquid
- Advantages:
  - Fast
  - Acceptable accuracy
- Disadvantages:
  - Doesn’t account for tube circuit configuration
  - Doesn’t account for the effect of successive rows

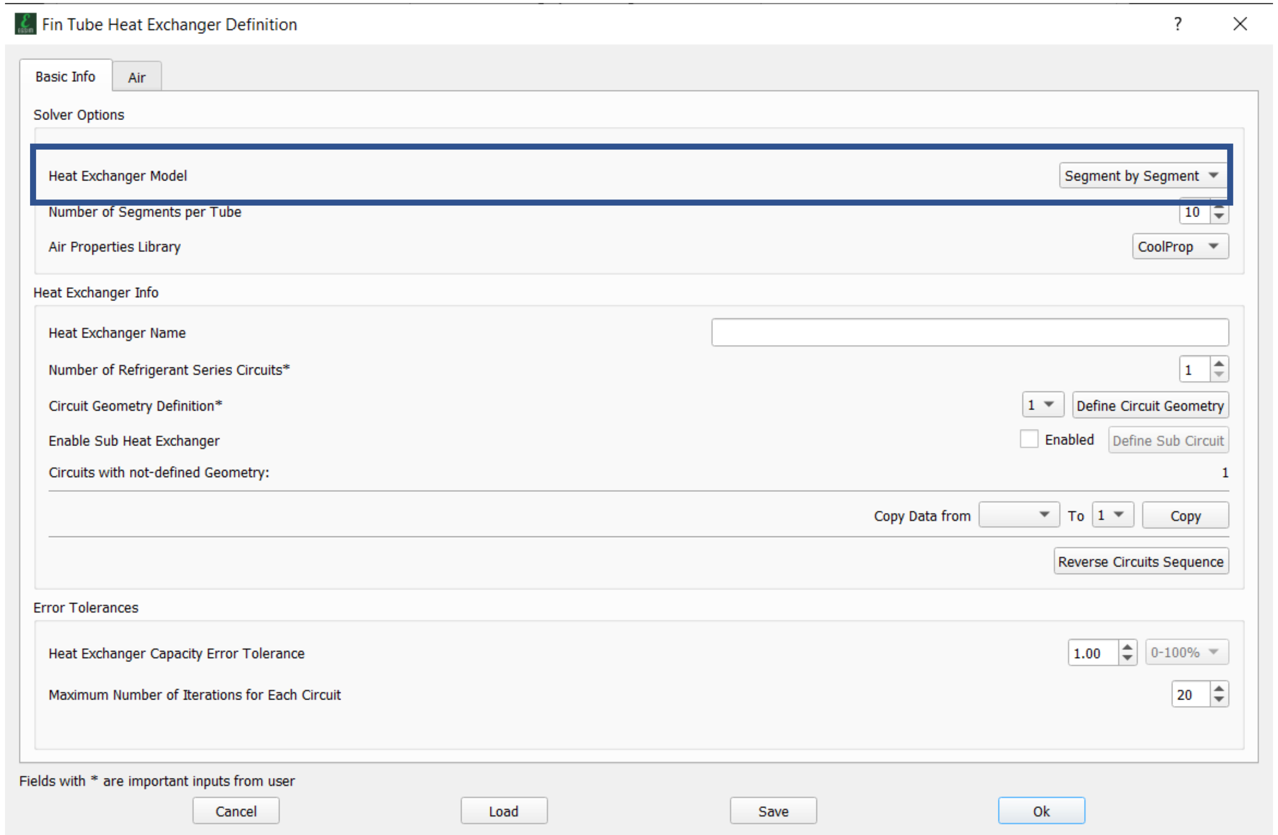
Segment-by-Segment Solver

- Each tube will be divided to **Number of Segments per Tube** defined by the user
- Each segment will be treated as a separate heat exchanger and solved



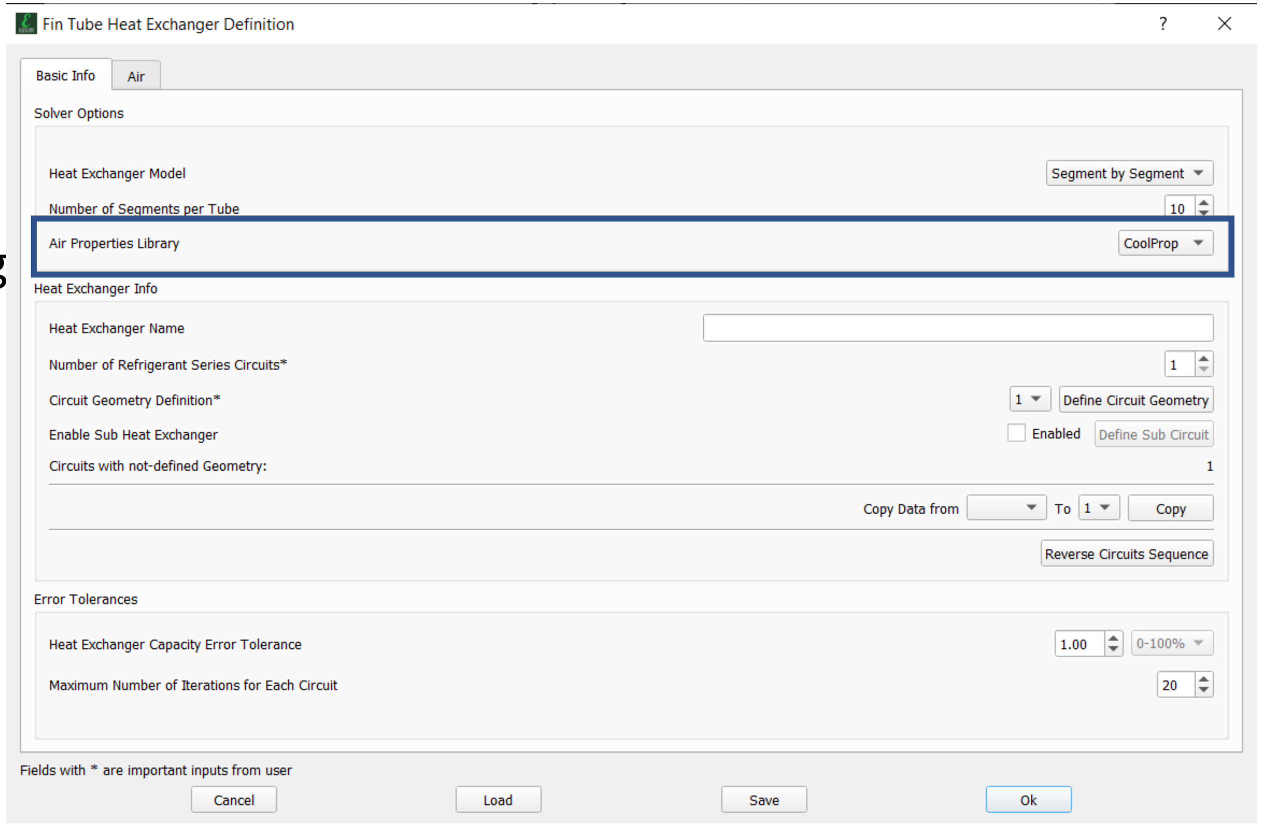
Segment-by-Segment Solver

- Advantages:
  - Highly accurate
  - Capable of evaluating the detailed tube circuit effects and successive tube rows
- Disadvantage: adding computational time, and hence slowing the overall simulation time



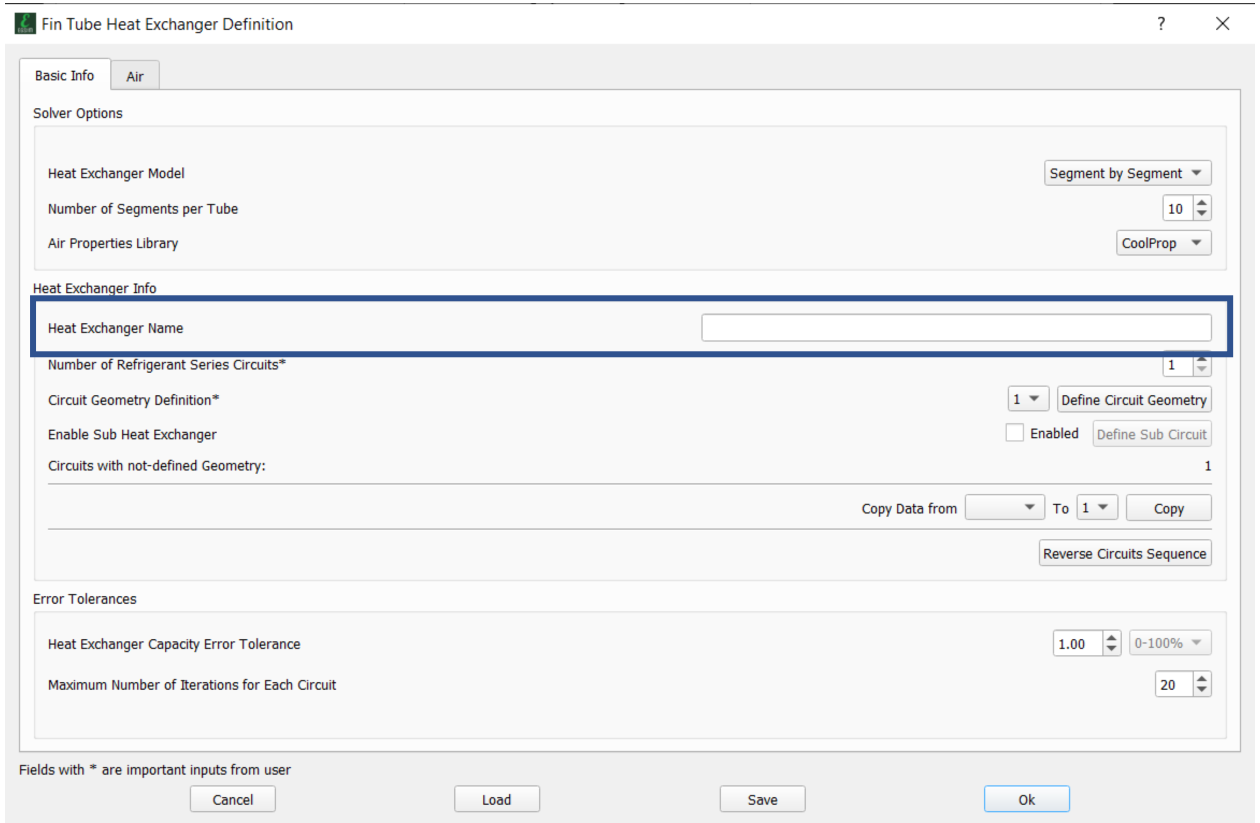
Air Properties Library

- The **Air Properties Library** defines the air properties calculations approach
  - **Coolprop** has the advantage of being accurate, reliable but relatively slow
  - **Psychrolob** uses ASHRAE approximations for air properties, which makes it faster to get air properties
- We advise the user to choose CoolProp



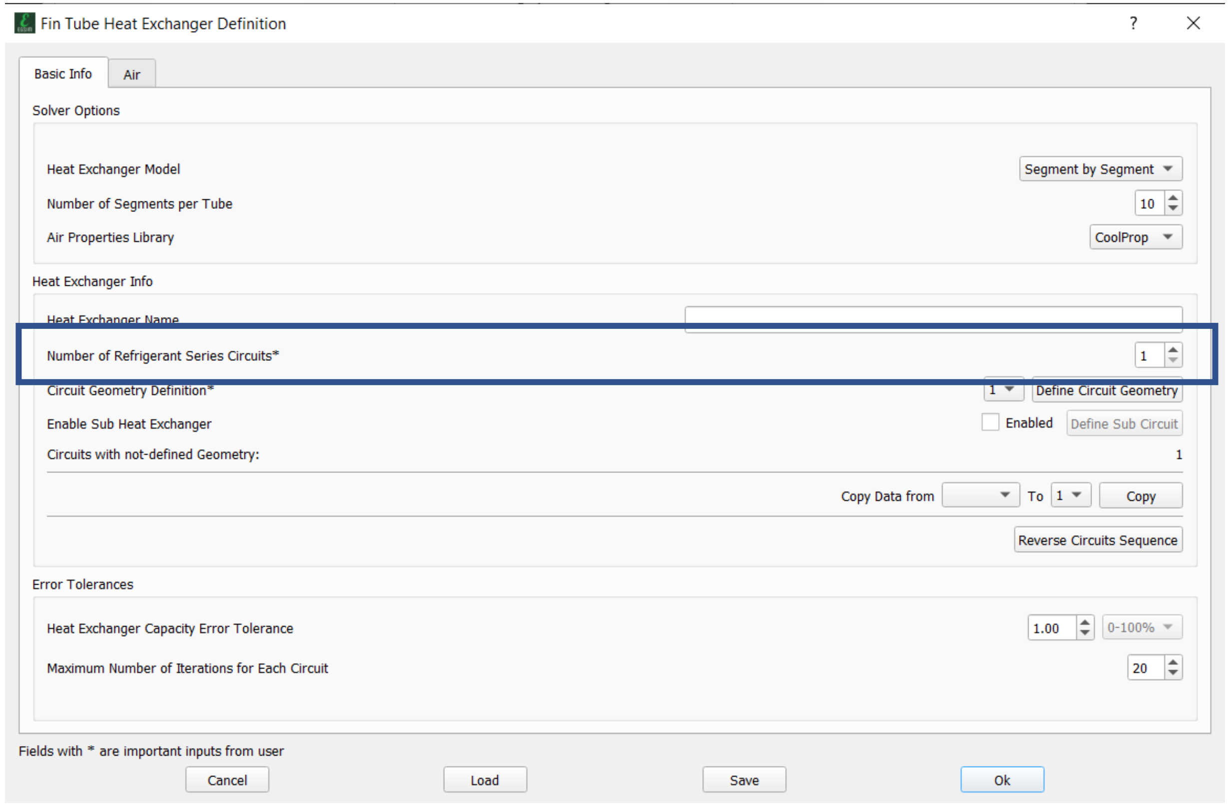
Heat Exchanger Name

- The **Heat Exchanger Name** is the name used to store the fin-tube heat exchanger into the database



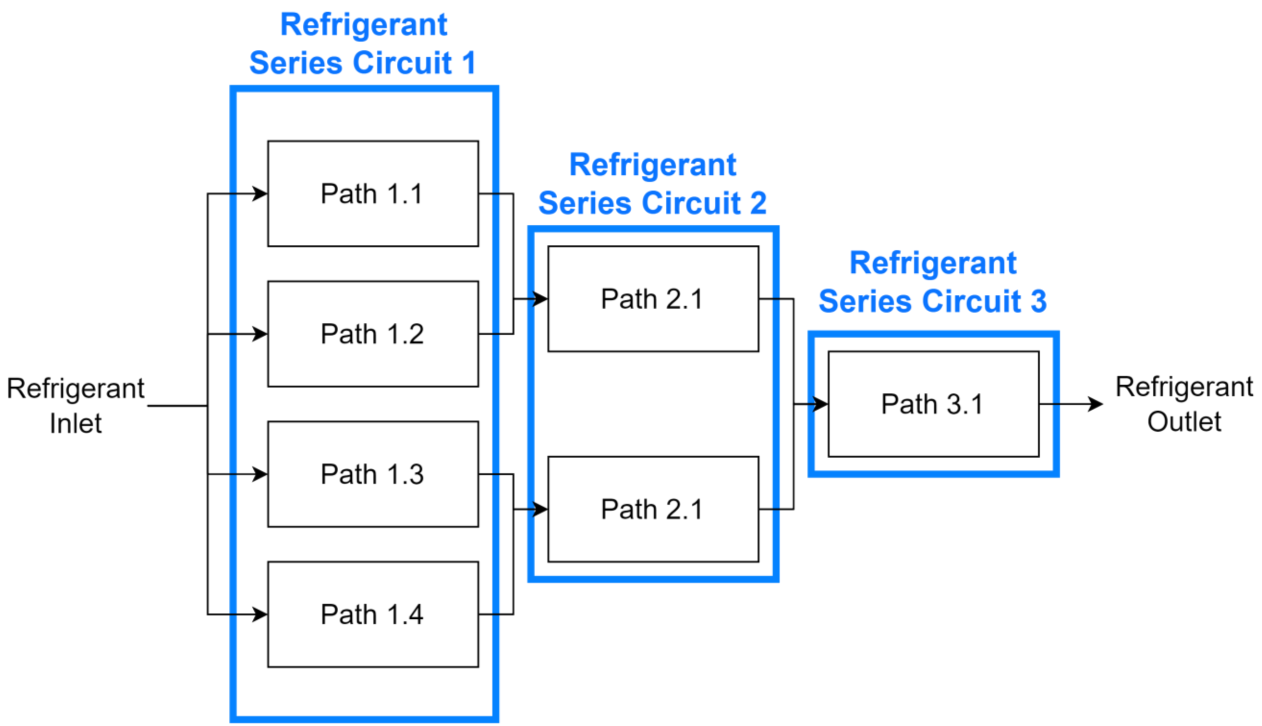
Refrigerant Circuits

- The **Number of refrigerant series circuits** defines the number of series (successive) circuits that the refrigerant will pass through from the inlet to the outlet of the heat exchanger
- Each refrigerant circuit may have multiple similar parallel paths



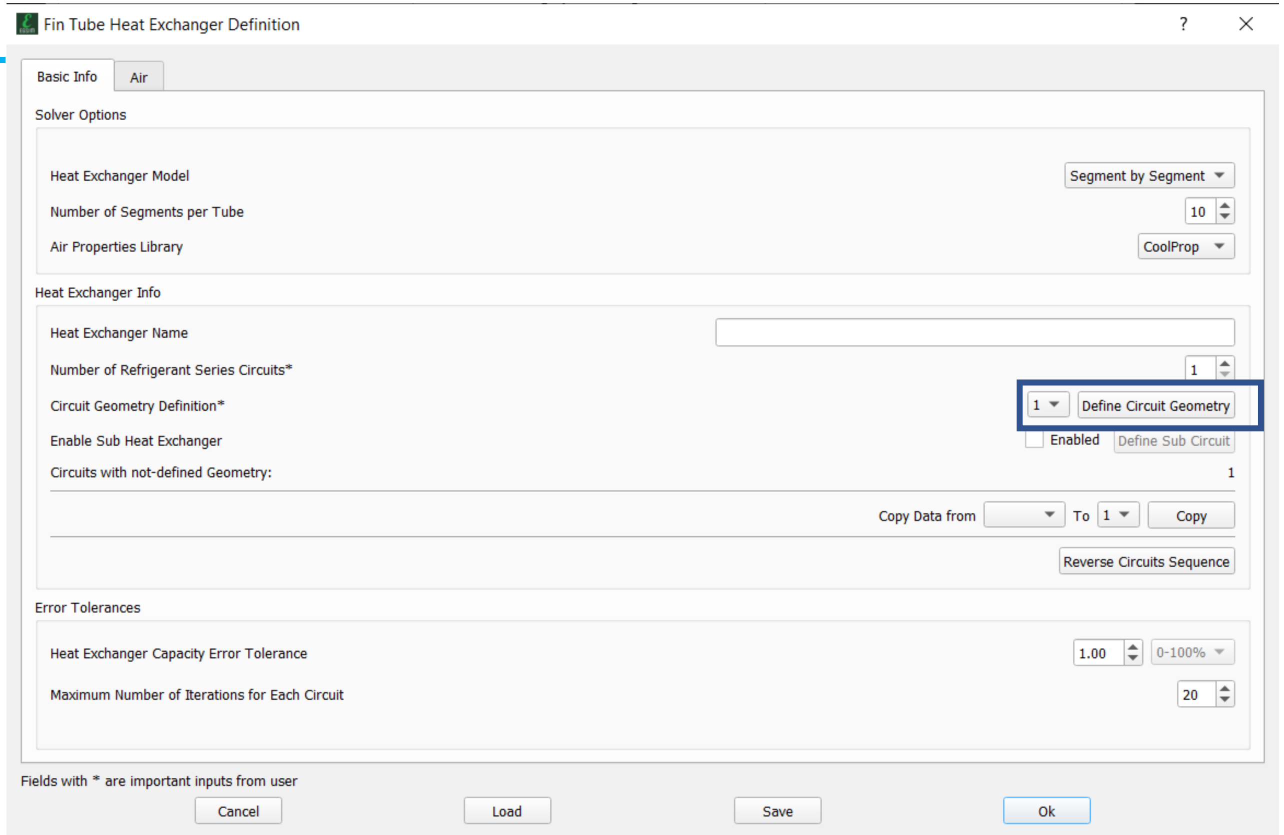
Refrigerant Circuits: Series vs. Parallel

- The following example shows a typical condenser circuitry
- In the following example:
  - The refrigerant enters series circuit 1, and passes through 4 parallel paths
  - Then The refrigerant enters series circuit 2, and passes through 2 parallel paths
  - Finally, the refrigerant enters series circuit 3, where it passes though 1 path



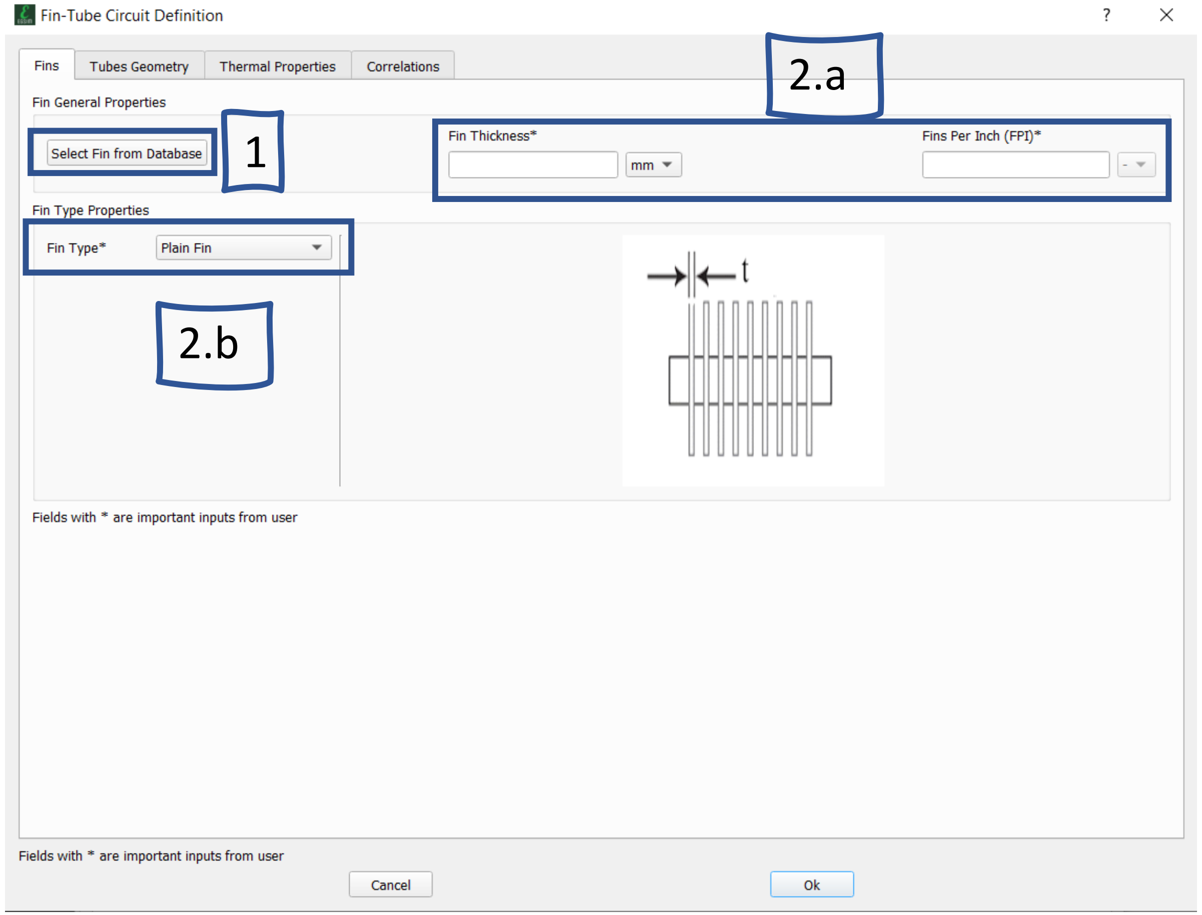
Series Circuit Definition

- For each series circuit, the user has to define the geometric design for each parallel path, and the number of paths which is called “**Number of duplicate parallel circuits**”
- First, the user selects which series circuit to work on from the **dropdown** list and then click on **Define circuit Geometry**



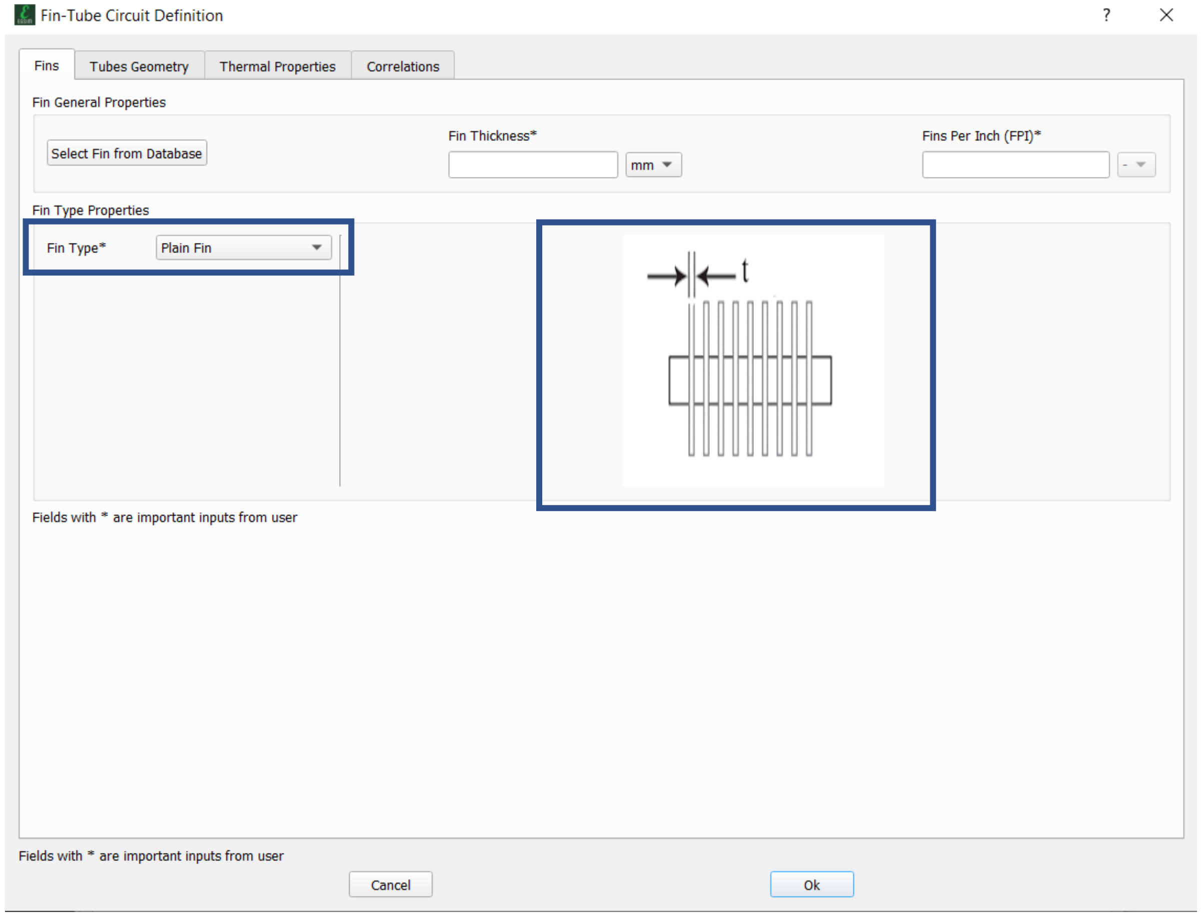
Series Circuit Geometry: Fins

- The user can either select a fin from the database or enter the detailed fin geometry
  1. Predefined fins can be selected by clicking on “**Select Fin from Database**”
  2. User-defined fins:
     a. Enter the values of **Fin Thickness** and **Fins Per Inch**
     b. Choose the **Fin Type** from the dropdown list (*the selected fin-type geometry will be demonstrated on the right*)



Series Circuit Geometry: Fins, cont.

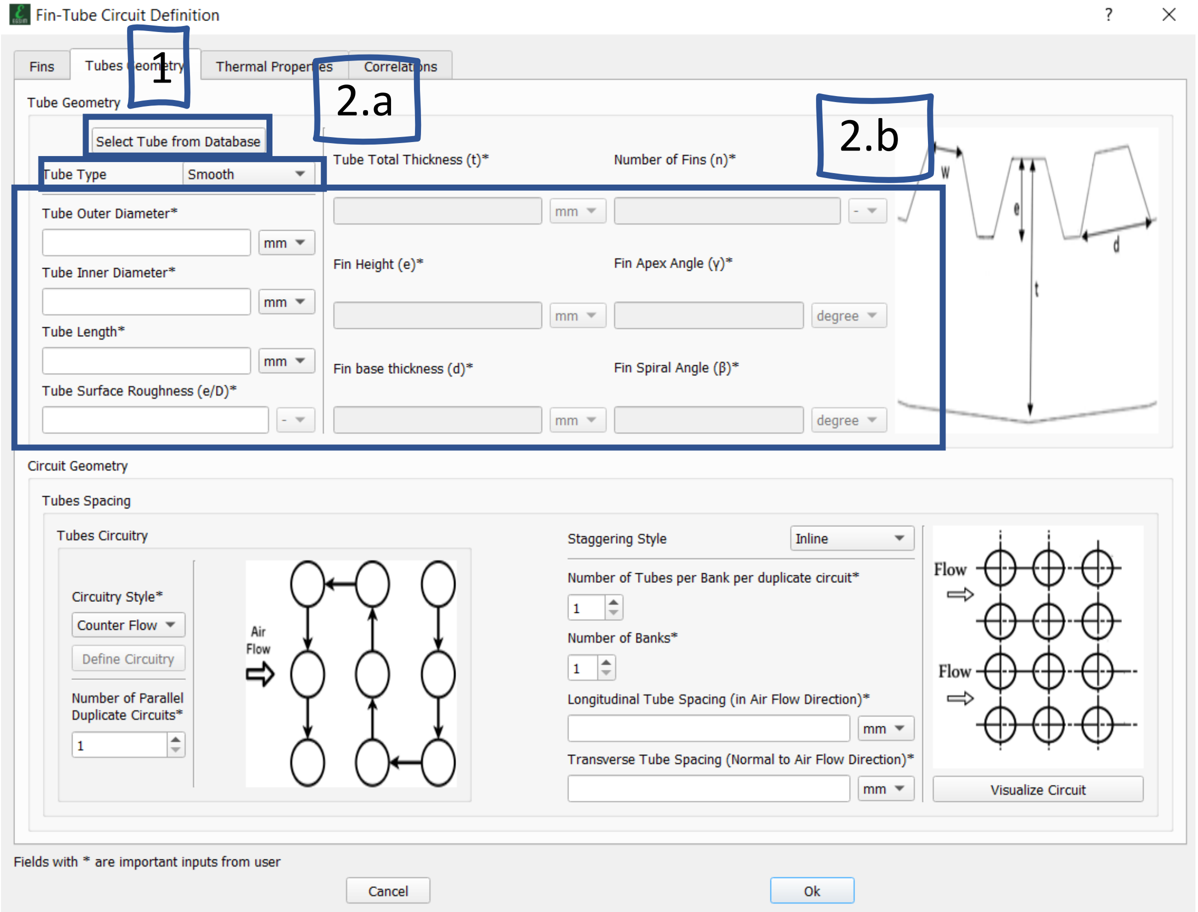
- Supported fin types are:
  - Plain fins
  - Wavy (Herringbone) fins
  - Wavy-Louvered fins
  - Louvered Fins
  - Slit fins
- For each fin type, additional parameters of the fin geometry might be needed, as defined in the illustration shown on the right



Series Circuit Geometry: Tubes Geometry

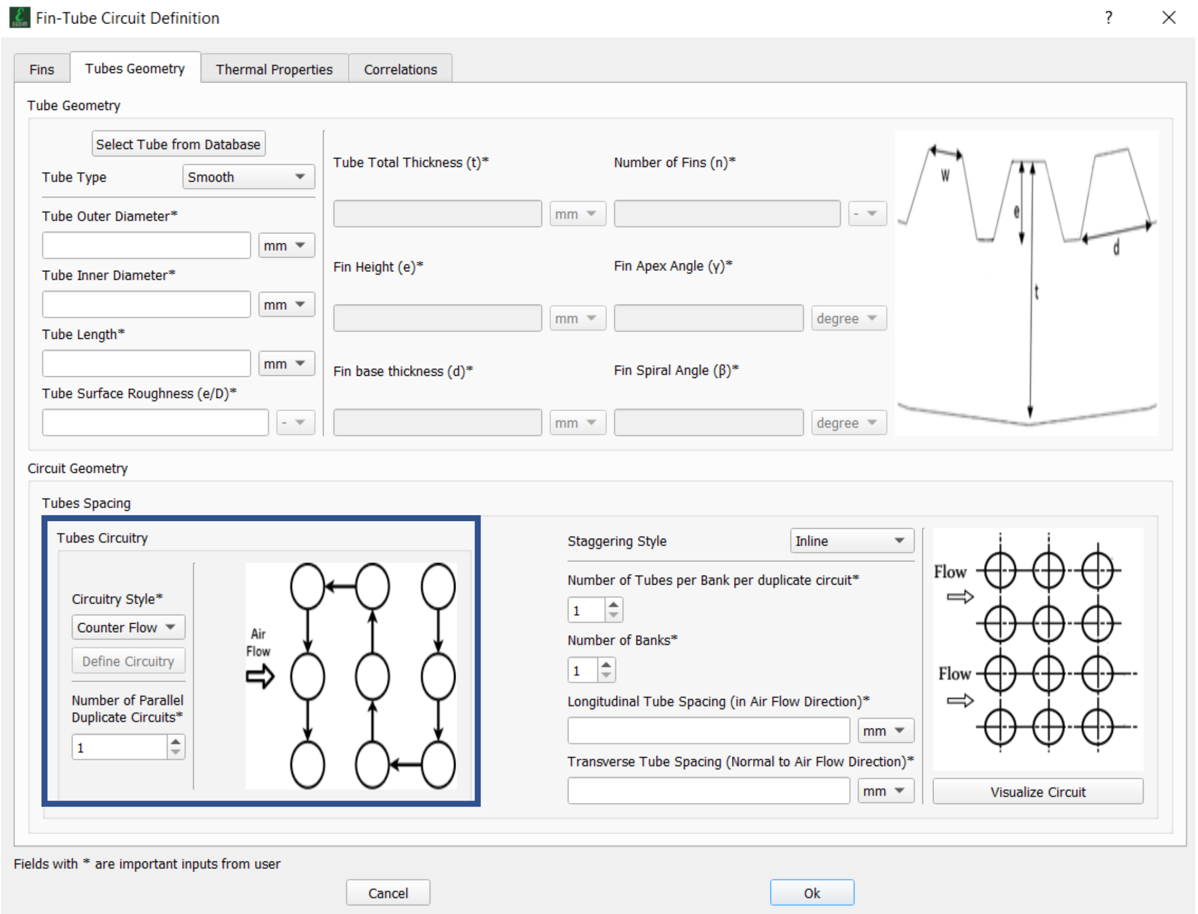
- The user can either select a tube from the database or enter the detailed tube geometry
    1. Predefined tubes can be selected by clicking on "**Select Tube from Database**"
    2. User Defined Tubes:
        a. The user has to first select the **Tube Type** (*either smooth tubes or micro-finned/internally grooved*)
        b. For each type, relevant geometric parameters will have to be input by the user

For both types, make sure you entered the correct tube length



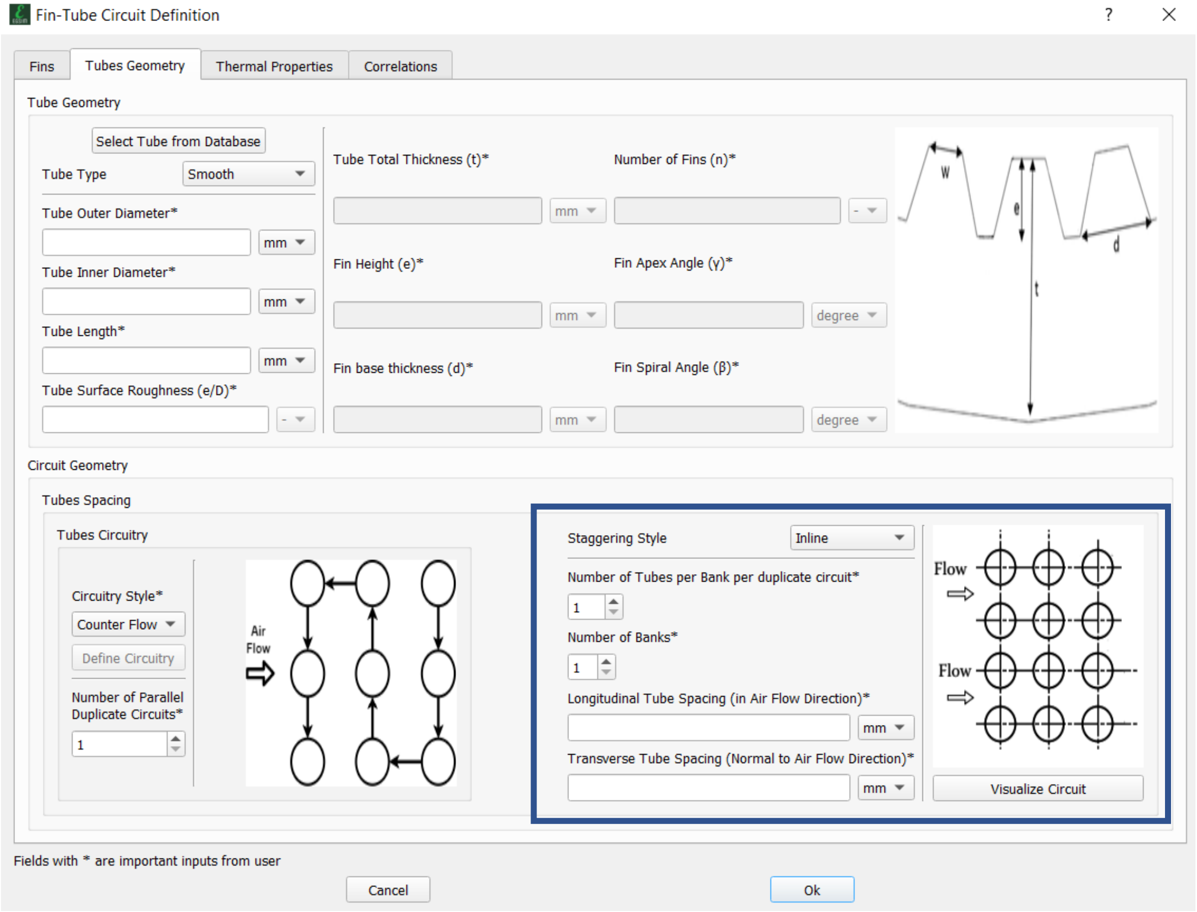
Series Circuit Geometry: Circuit Geometry

- The user has to specify the **Circuitry Style** (*available only for segment-by-segment solver*)
  - Counter Flow
  - Parallel Flow
  - Custom Circuitry
- The user also has to enter the **Number of Parallel Duplicate Circuits**



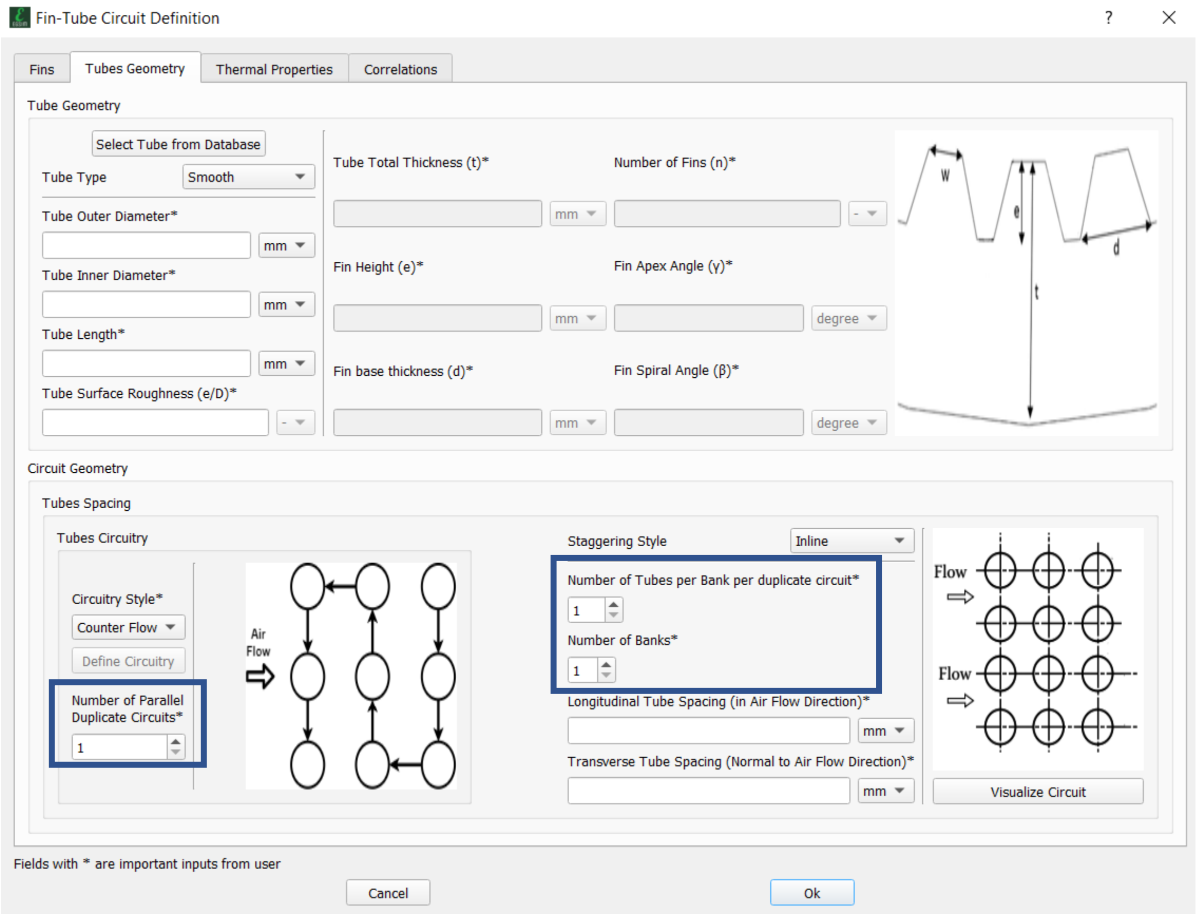
Series Circuit Geometry: Circuit Geometry

- The user then defines the tubes Staggering Style (*only if the segment-by-segment solver is used*); *the illustration on the right will depict the selected option*
- The user also enters the number of tubes per bank (row) per duplicate circuit (path); this defines the number of vertical tubes in this path
- The user also enters the number of tubes per bank (row) per duplicate circuit; this defines the number of horizontal tubes in this path



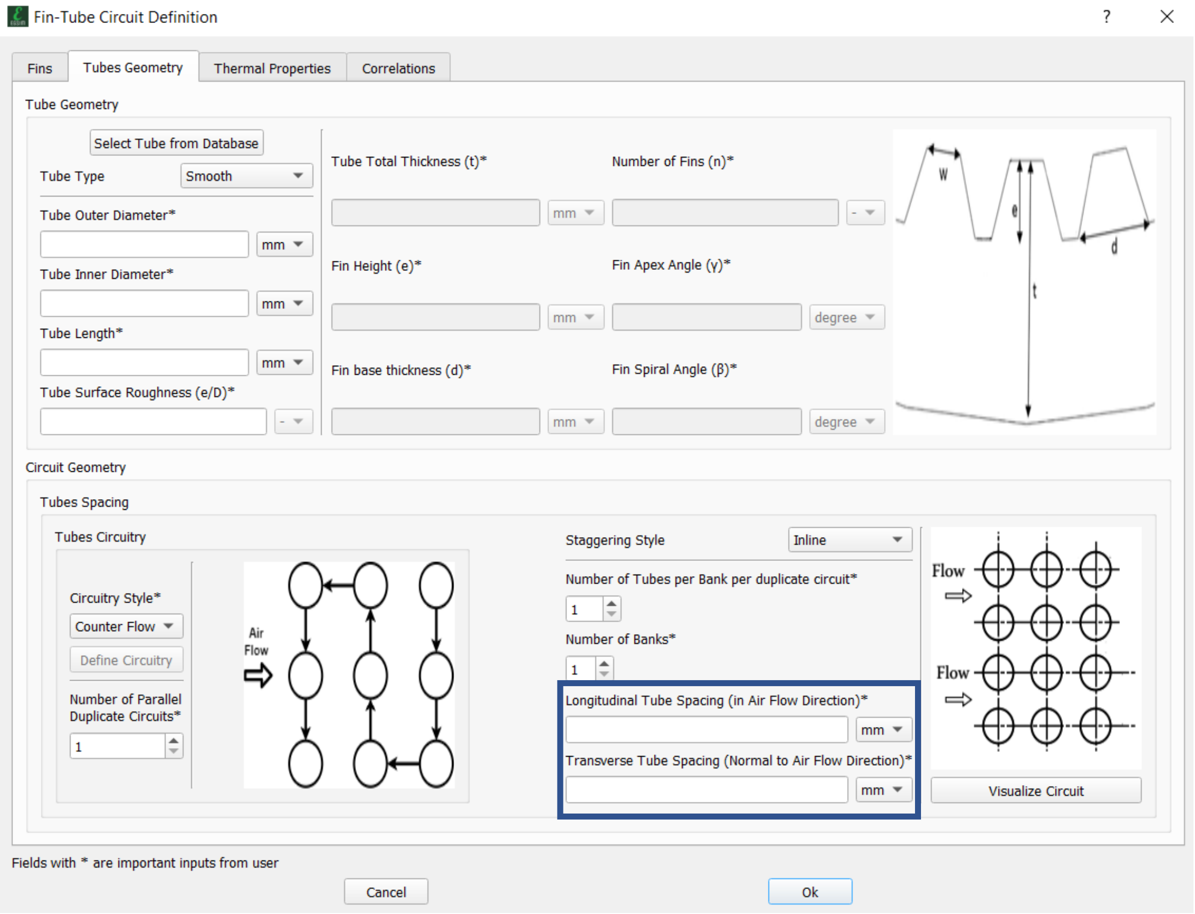
Series Circuit Geometry: Circuit Geometry, cont.

- The total number of tubes of this path, is defined by the multiplication of “**Number of Tubes per Bank per Duplicate circuit**” and “**Number of Banks**”



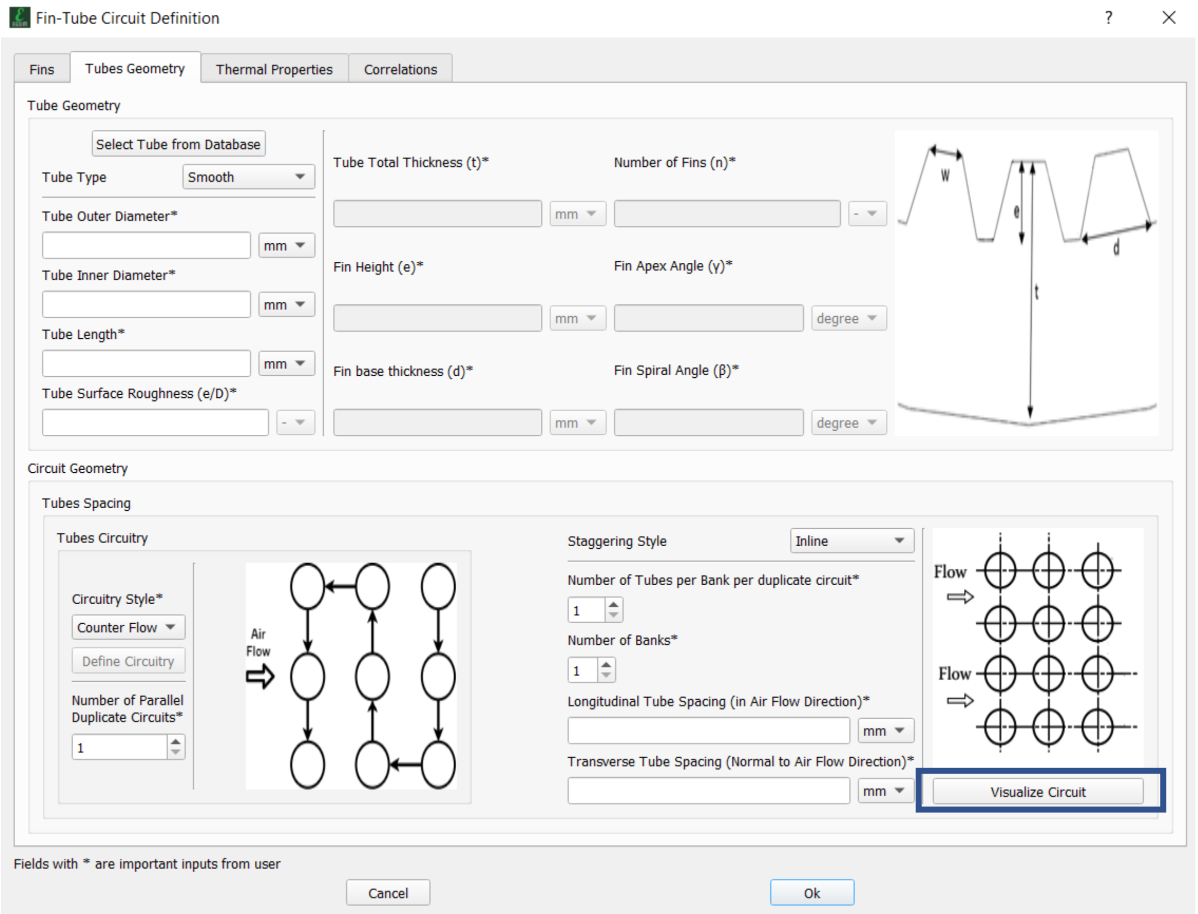
Series Circuit Geometry: Circuit Geometry, cont.

- The user has to define the **Longitudinal Tube Spacing** which is the distance between two successive horizontal tubes (in the air flow direction)
- The user has to define the **Transverse Tube Spacing** which is the distance between the center of two successive vertical tubes (normal to air flow direction)



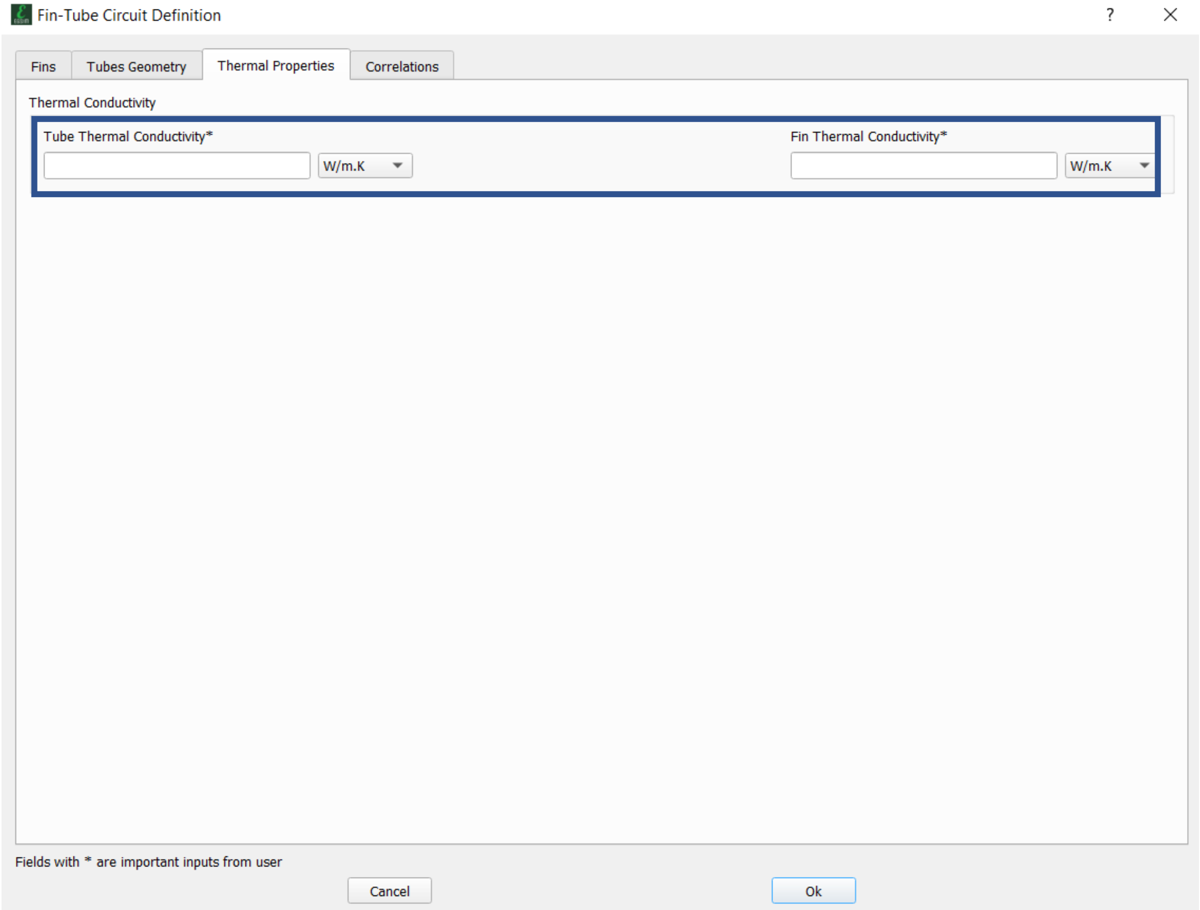
Series Circuit Geometry: Circuit Geometry, cont.

- You can visualize the circuit using "**Visualize Circuit**" button to make sure all inputs are correct



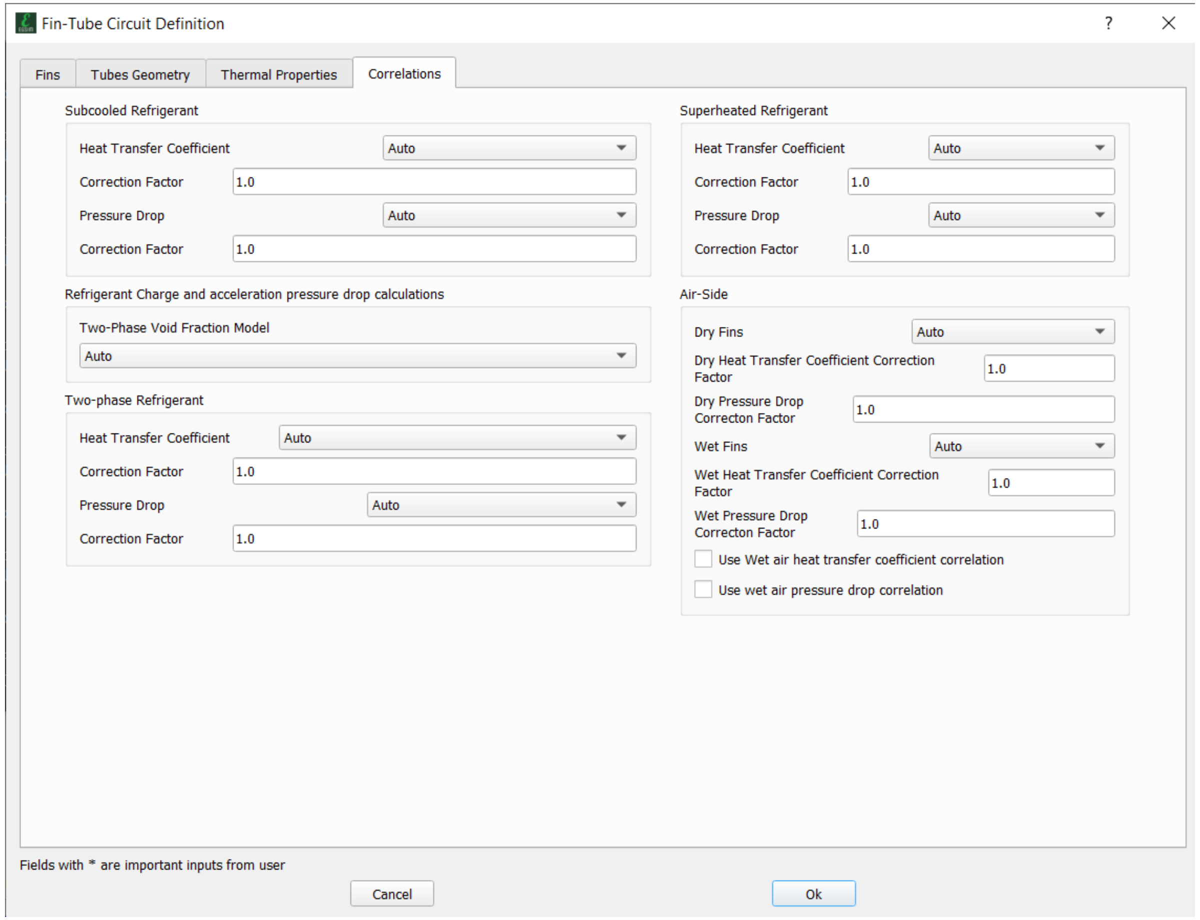
Series Circuit Geometry: Thermal Properties

- In this tab, the user defines the **Tube Thermal Conductivity** and the **Fin Thermal Conductivity**
- Both SI and IP units are available in the dropdown list.



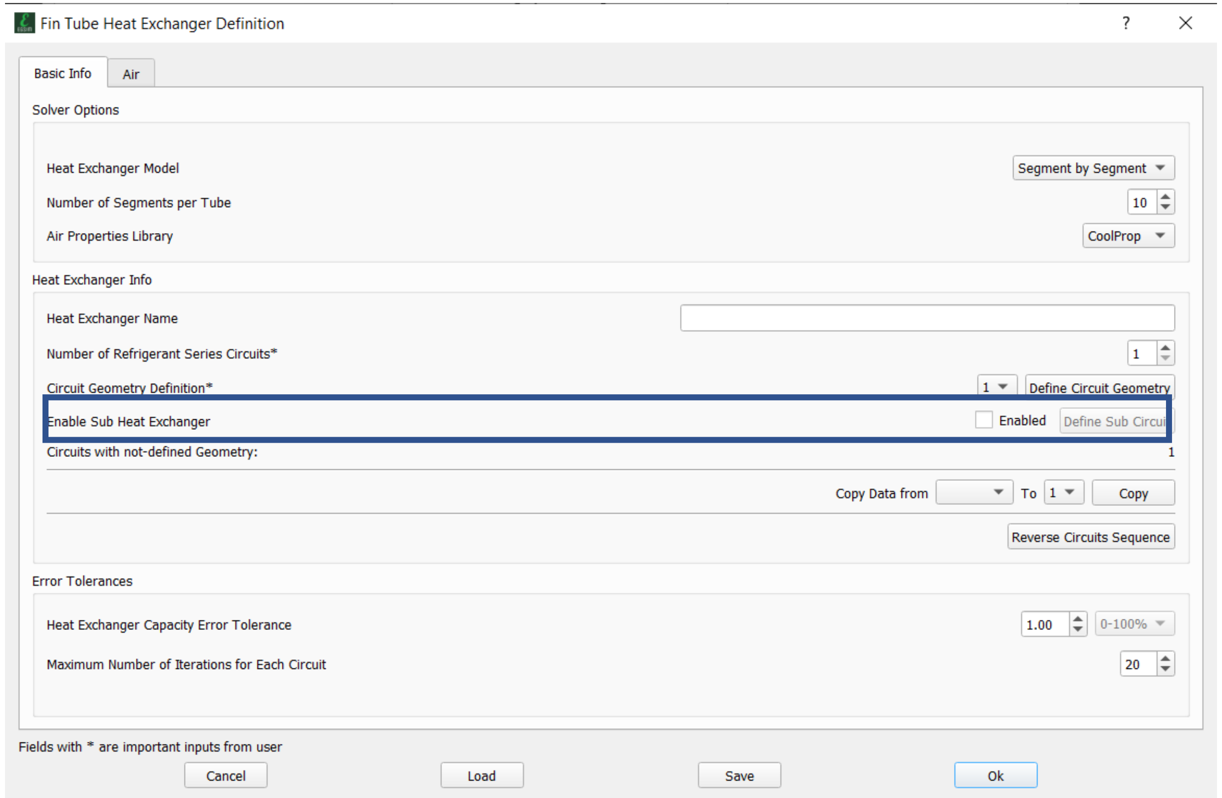
Series circuit Geometry: Correlations

- On this tab, the user can define correlations to use for the evaluation of heat transfer and pressure drop
    - Refrigerant side **Subcooled**, **Two-phase**, and **Superheated**
    - Air-side; dry and wet
- Selecting "**Auto**" will allow the software to choose the most suitable correlation according to a smart algorithm which takes the geometric and physical conditions of the case into account
- **Correction Factors** can be used to provide needed correction to the heat transfer and pressure drop evaluation for validation purposes



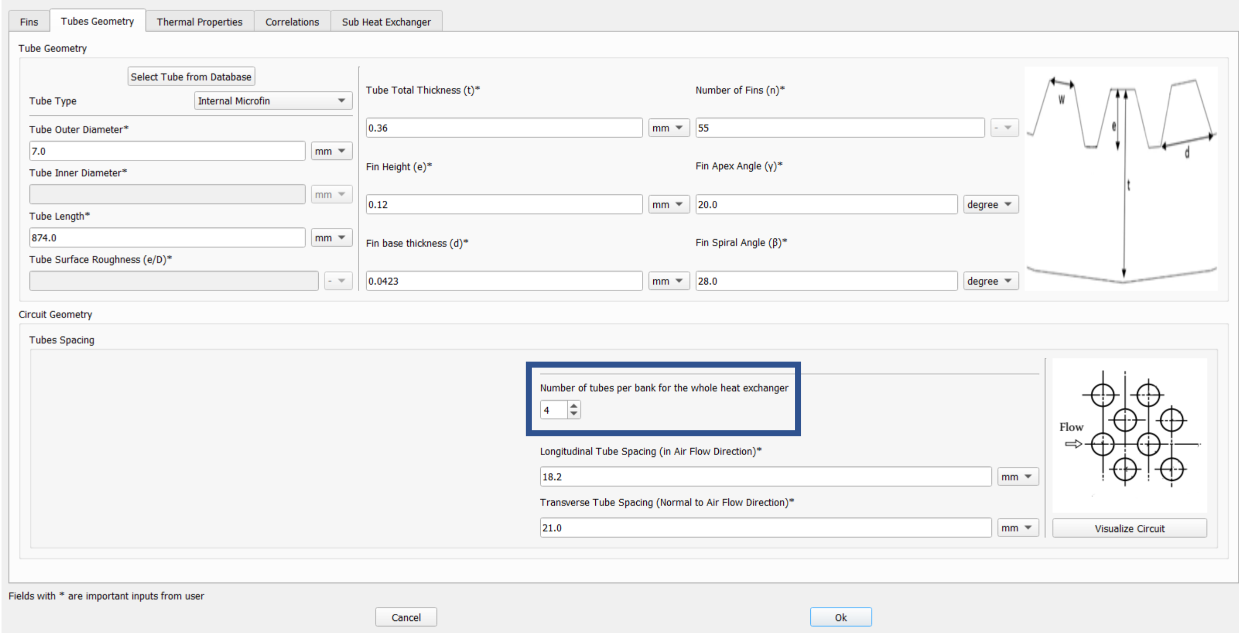
Sub Heat Exchanger

- This model is used to simulate the performance when a truncated bank is added either before the first row or after the last row
- This can be activated by checking the "**Enabled**" box
- To define the sub heat exchanger, click on "**Define Sub Circuit**"



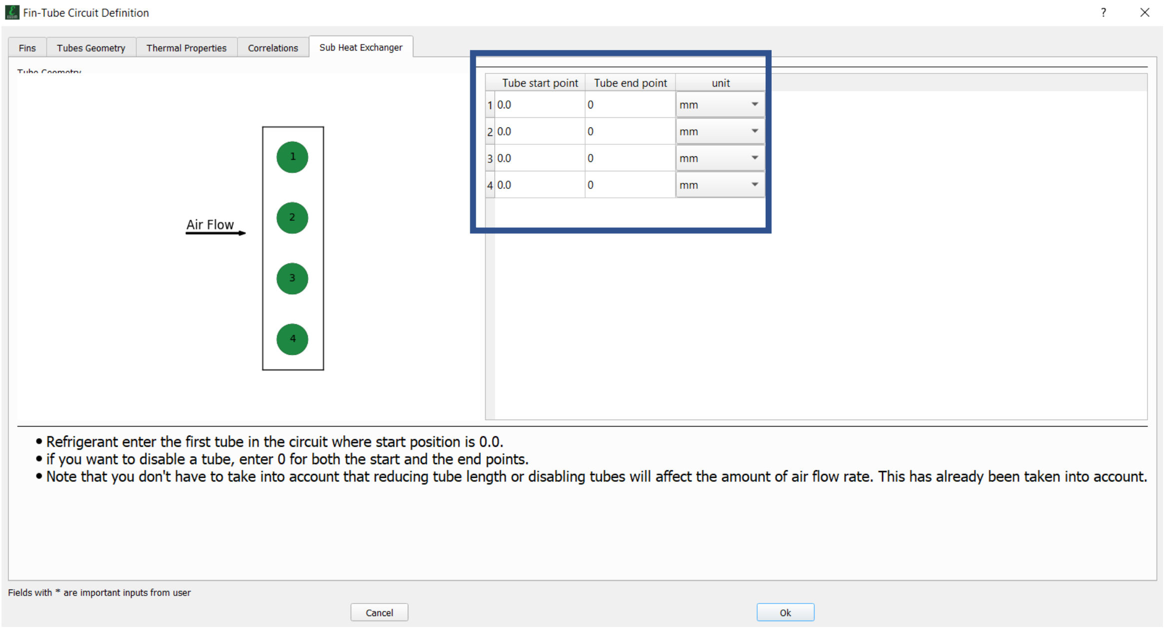
Sub Heat Exchanger, cont.

- The definition of the sub circuit is similar to the series circuit geometry definition, except for a new tab which contains the parameters of the sub circuit
- In the **Tube Geometry** tab, the user has to enter the **Total Number of Tubes per Bank for the whole heat exchanger**; this value should be equal to the total number of vertical tubes used in the heat exchanger (normal to airflow direction)
- The user will be able to remove tubes in the **Sub Heat Exchanger** tab



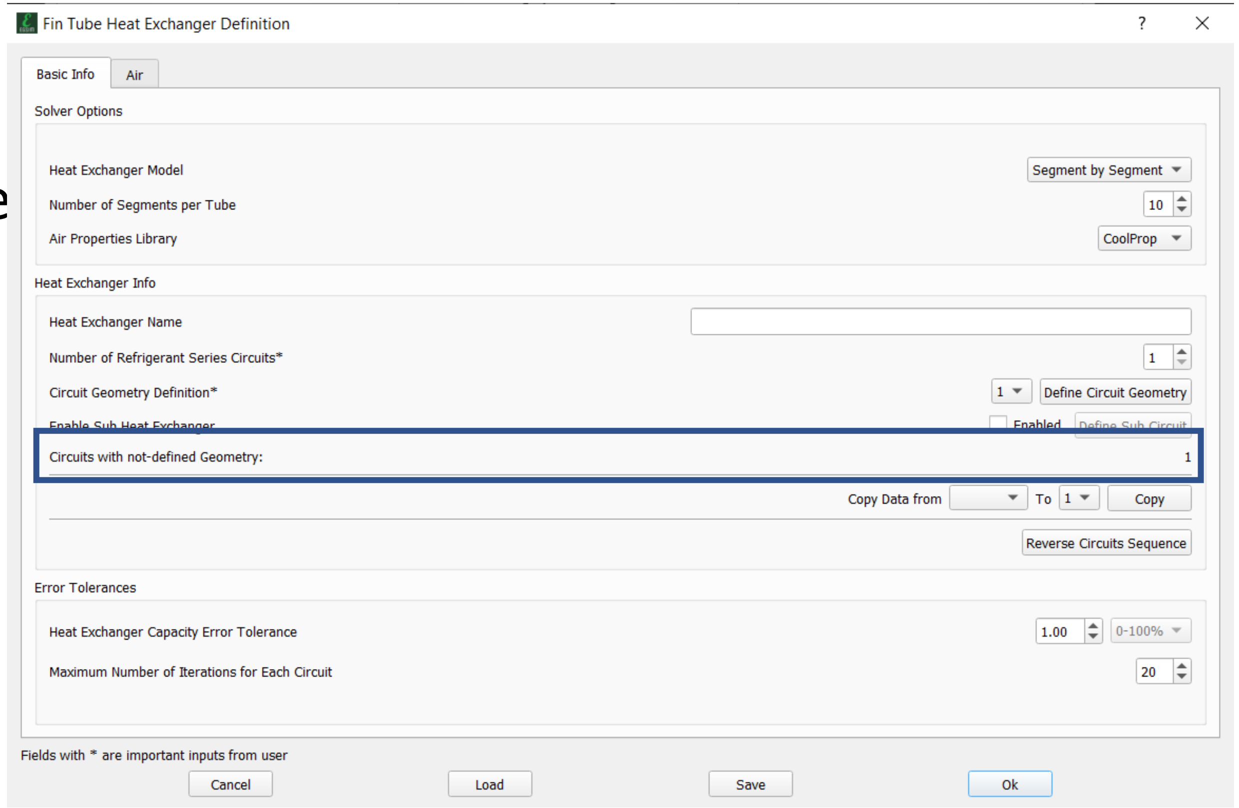
Sub Heat Exchanger, cont.

- In the **Sub heat exchanger** tab, for each tube in the sub heat exchanger the user can
  - Remove a part of a tube by defining the start and end points
  - Remove an entire tube by entering a value of 0 for the start and end points



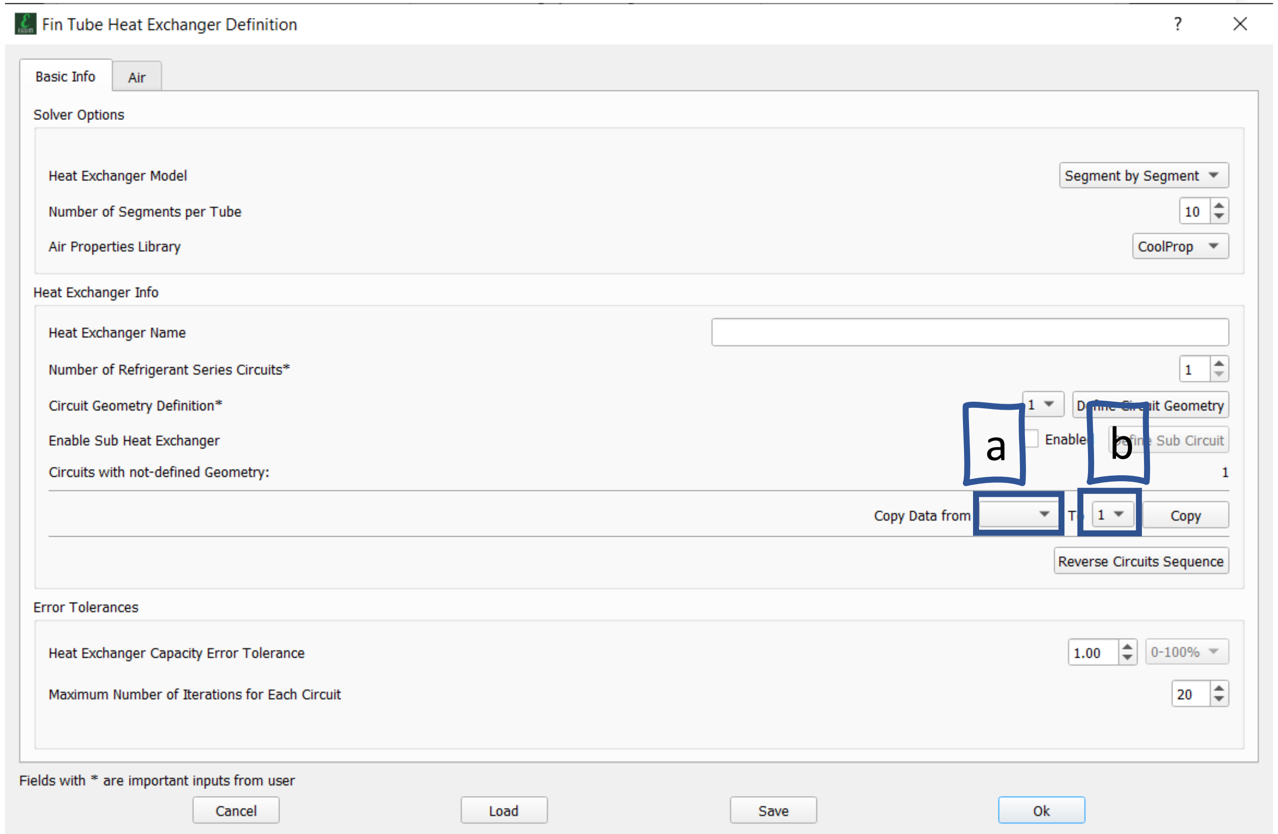
Circuits with Not-Defined Geometry

- The user will be able to track the progress of circuit definition as the list of non-defined circuits will appear next to **Circuits with not-defined Geometry**
- The heat exchanger cannot be saved before finishing the geometry definition for all circuits



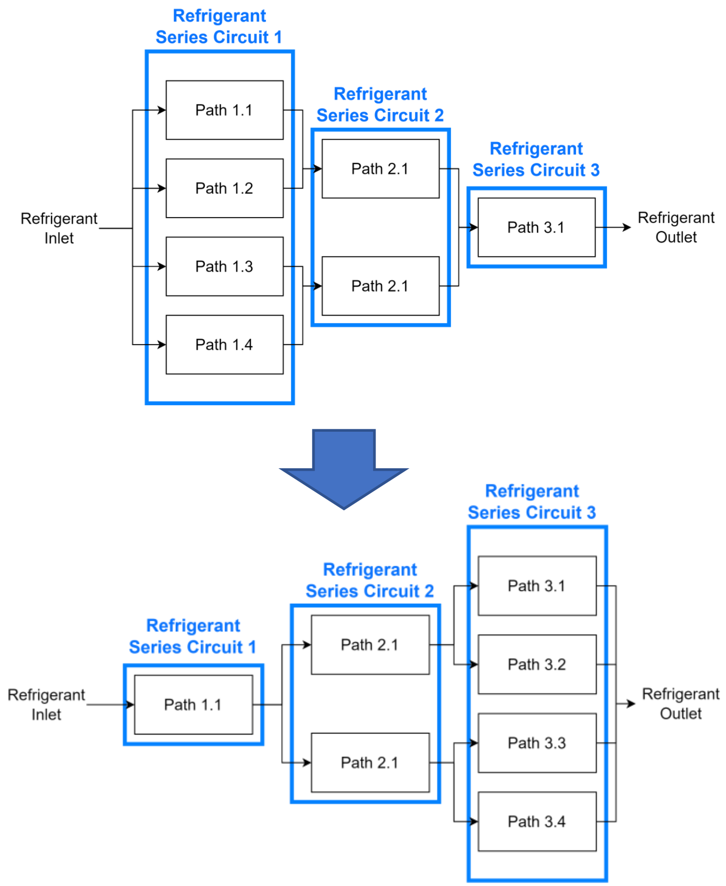
Copying Data Between Circuits

- The user can copy data from a defined series circuit to another series circuit directly by selecting the most relevant fully-defined series circuit from box **a** and selecting the undefined circuit in box **b** and then clicking **Copy**



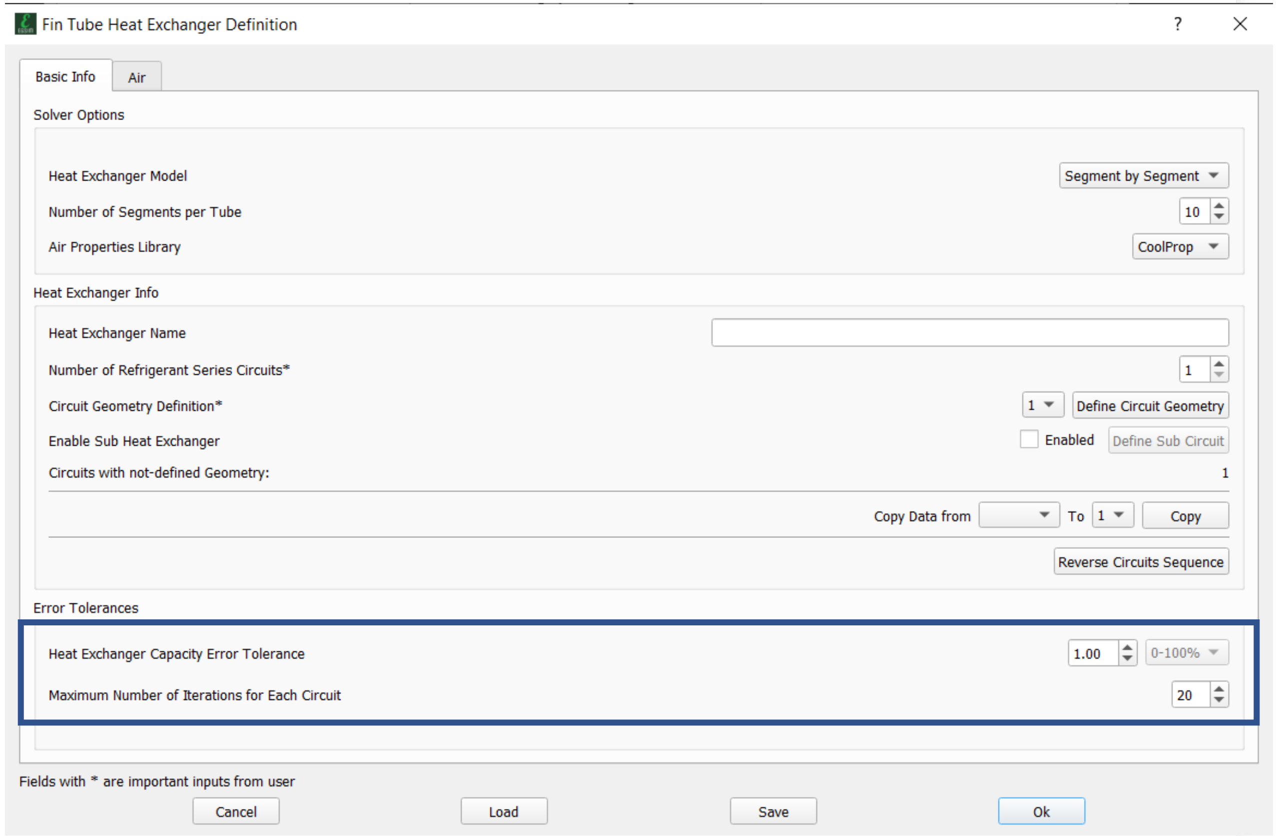
Reverse Circuits Sequence

- Clicking on **Reverse Circuits Sequence** will reverse the series circuits order
- This option is helpful when evaluating the performance of a heat pump as the refrigerant flow direction is typically switched between cooling and heating modes



Error Tolerances

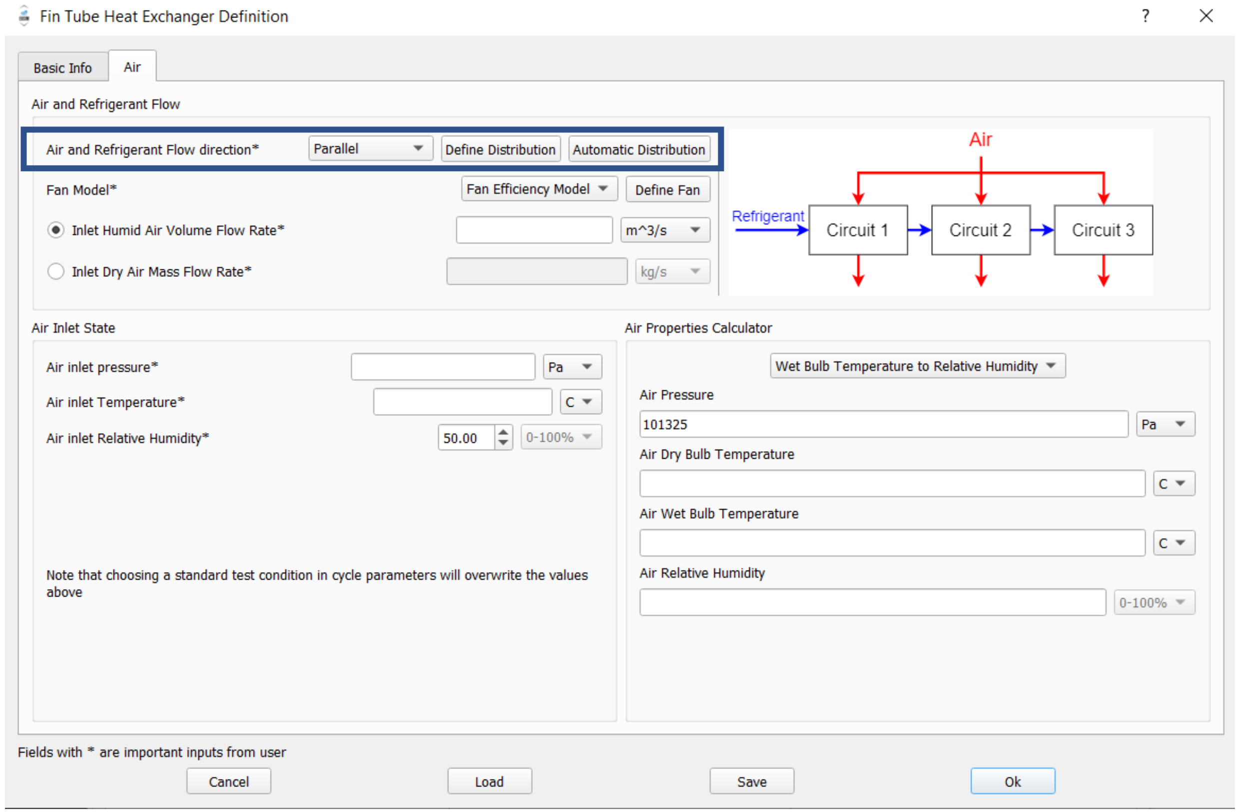
- Segment-by-Segment solver relies on trial-and-error, the solver convergence criteria is the difference in heat exchanger capacity
- The **Heat Exchanger Capacity Error Tolerance** defines the maximum allowed error
- The **Maximum Number of Iterations for Each Circuit** is used to limit the solver time, the user can select how many iterations can be reached before the solver exits even if the convergence criteria are not met



Air Tab

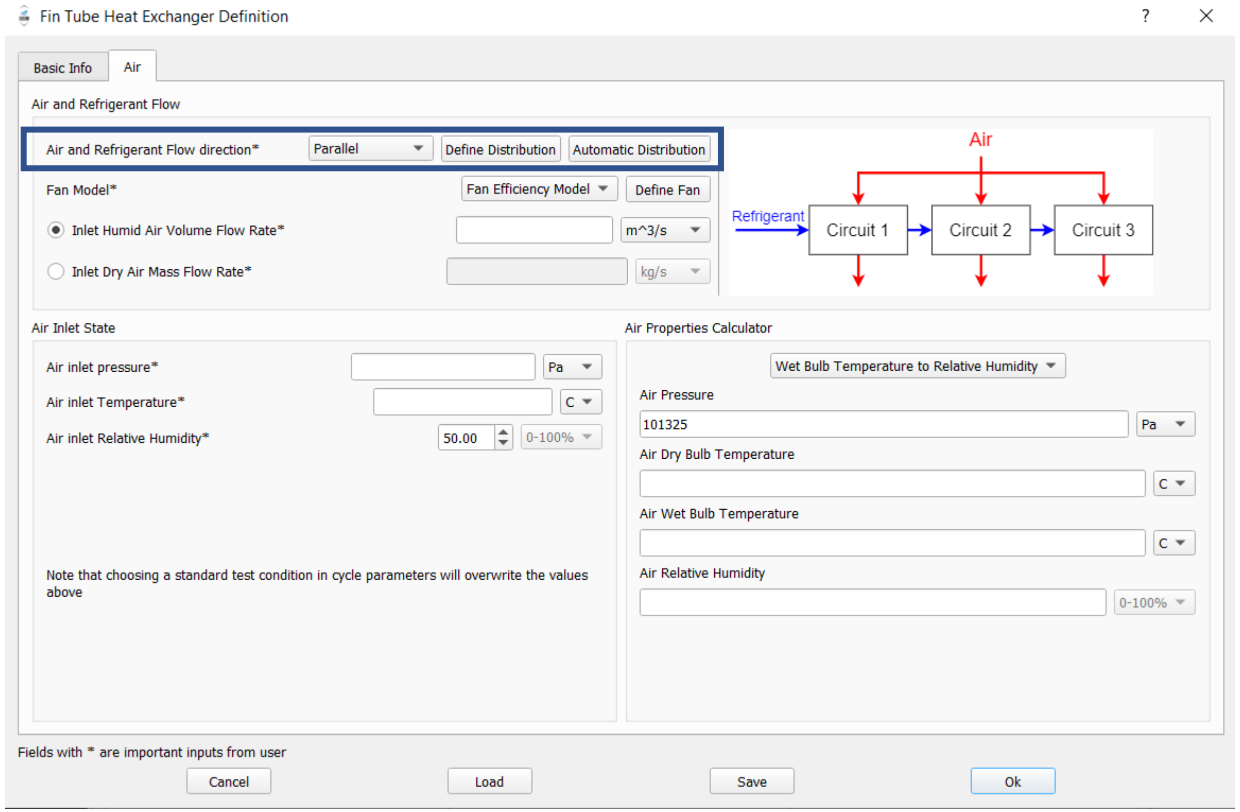
Air and Refrigerant Flow

- On this tab, the user can define the **Air and Refrigerant Flow Direction**
- In the illustration, refrigerant circuits represent the series circuits
- In case there is no sub-heat exchanger, 3 options will be available:
  - Parallel
  - Series-Parallel
  - Series-Counter



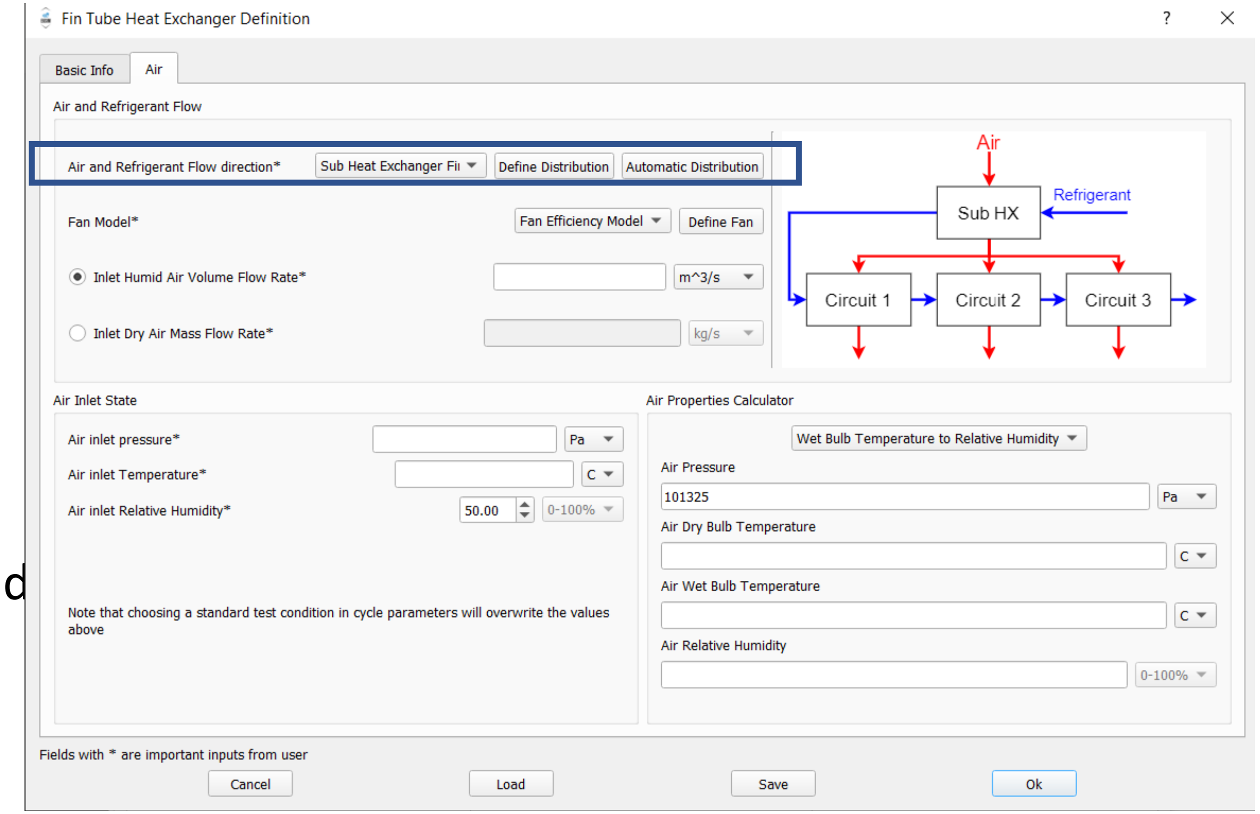
Air and Refrigerant Flow, cont.

- For "Parallel", you will need to define the air distribution ratio for each refrigerant series circuit
- If all refrigerant series circuits have the same number of banks (rows), then the user can click **Automatic Distribution** and the distribution ratios will be calculated depending on total number of vertical tubes in each series circuit



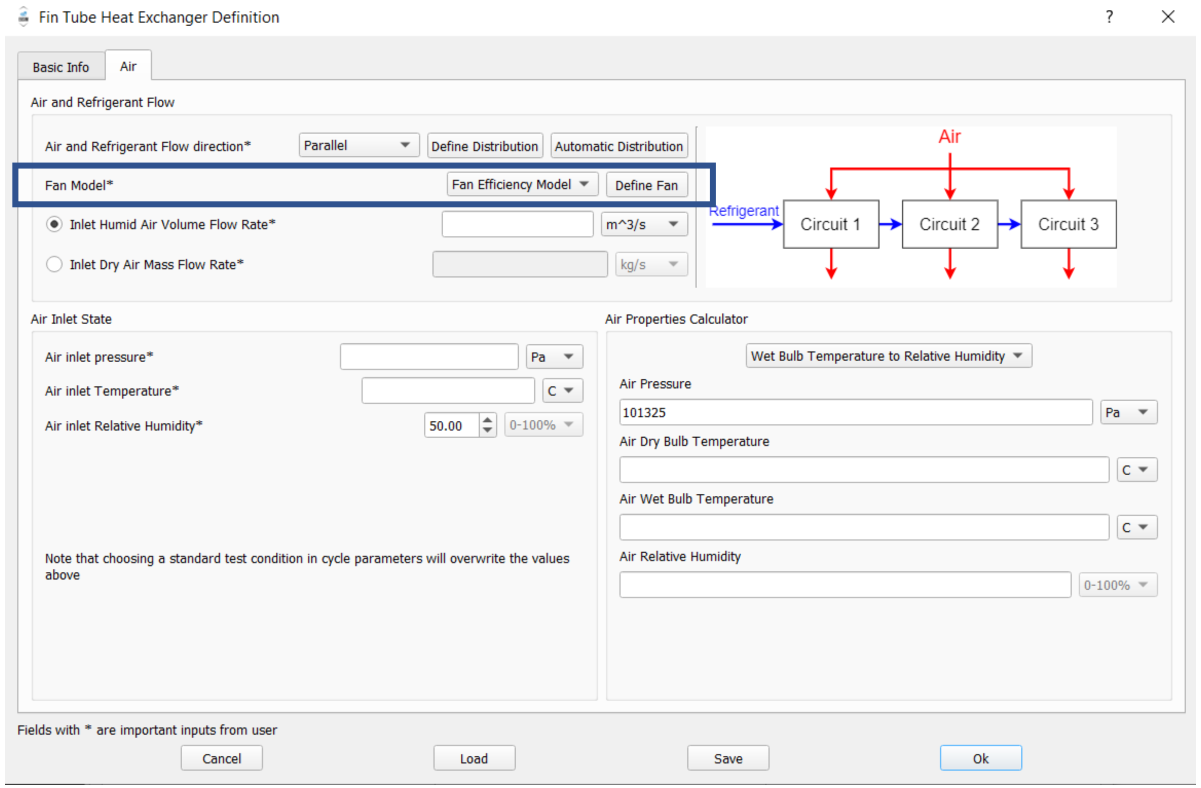
Air and Refrigerant Flow, cont.

- If sub-heat exchanger is enabled, the user will have 2 available options:
  - **Sub-heat exchanger first**
  - **Sub-heat exchanger last**
- The user will need to define the air distribution ratios for the refrigerant series circuits.
- If all refrigerant series circuits have the same number of banks (rows), then the user can click **Automatic Distribution** and the distribution ratios will be calculated depending on total number of vertical tubes in each series circuit



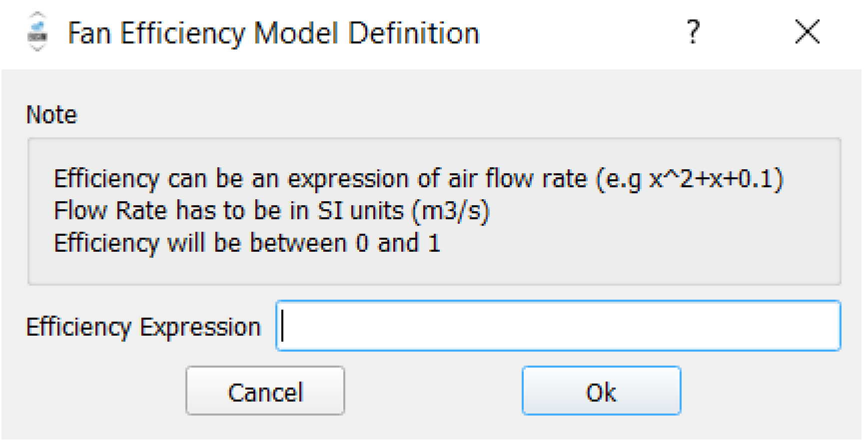
Fan Model

- The user can choose from 3 available fan models in EGSim:
  - **Fan efficiency model**
  - **Fan curve model**
  - **Fan Power model**



Fan Model: Fan Efficiency Model

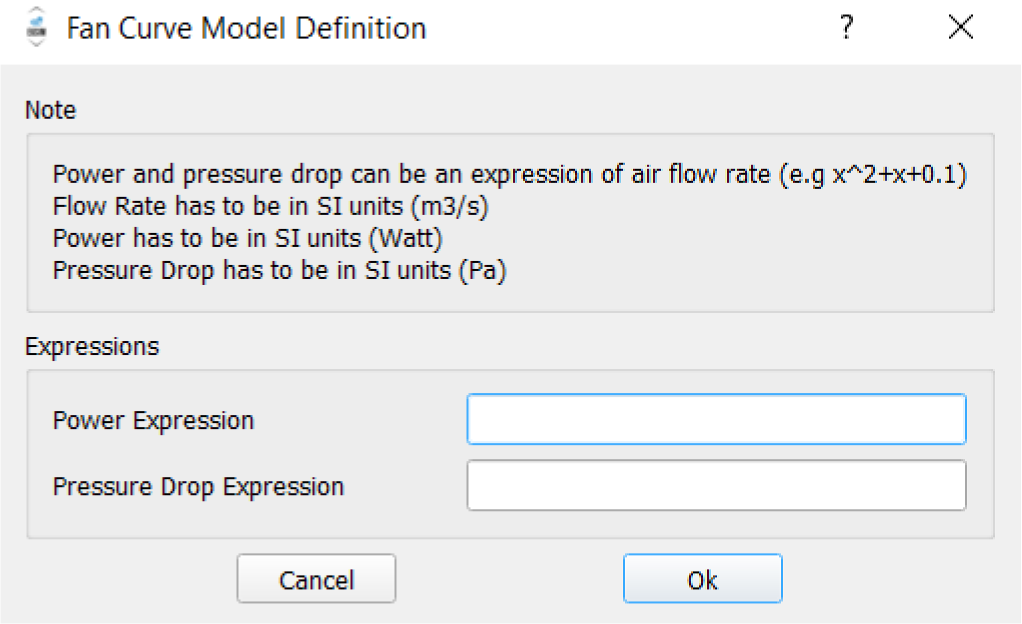
- In fan efficiency model, fan efficiency is defined as a function of the fan flowrate (in m^3/s)
- From fan efficiency and heat-exchanger air pressure drop, fan power will be calculated
- The user can define:
  - Any mathematical expression:
    -28.292*x^2 + 7.6344*x - 0.0183
  - A constant value: 0.6



Fan Model: Fan Curve Model

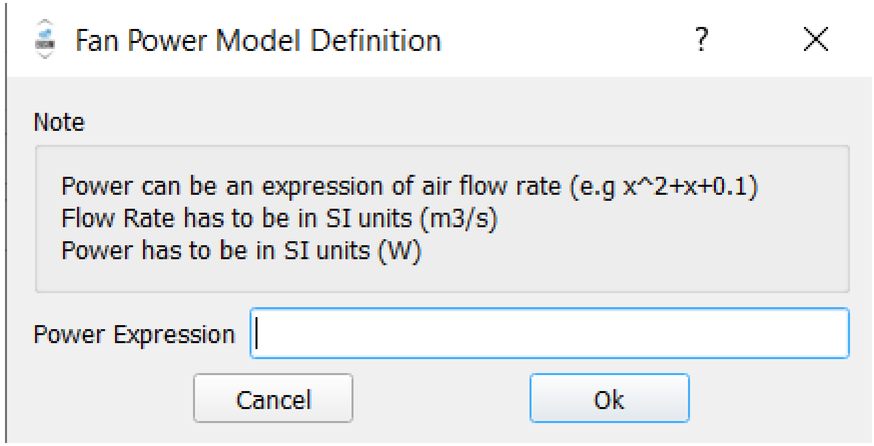
- In fan curve model, fan power and fan pressure-rise are defined as a function of fan flowrate (in m^3/s)
- Fan curve will be solved with heat-exchanger system curve to find operating air flowrate and corresponding fan power and efficiency
- The user can define:
  - Any mathematical expression:

    -1722.9*x^2 + 451.29*x + 1.5714
  - A constant value: 40



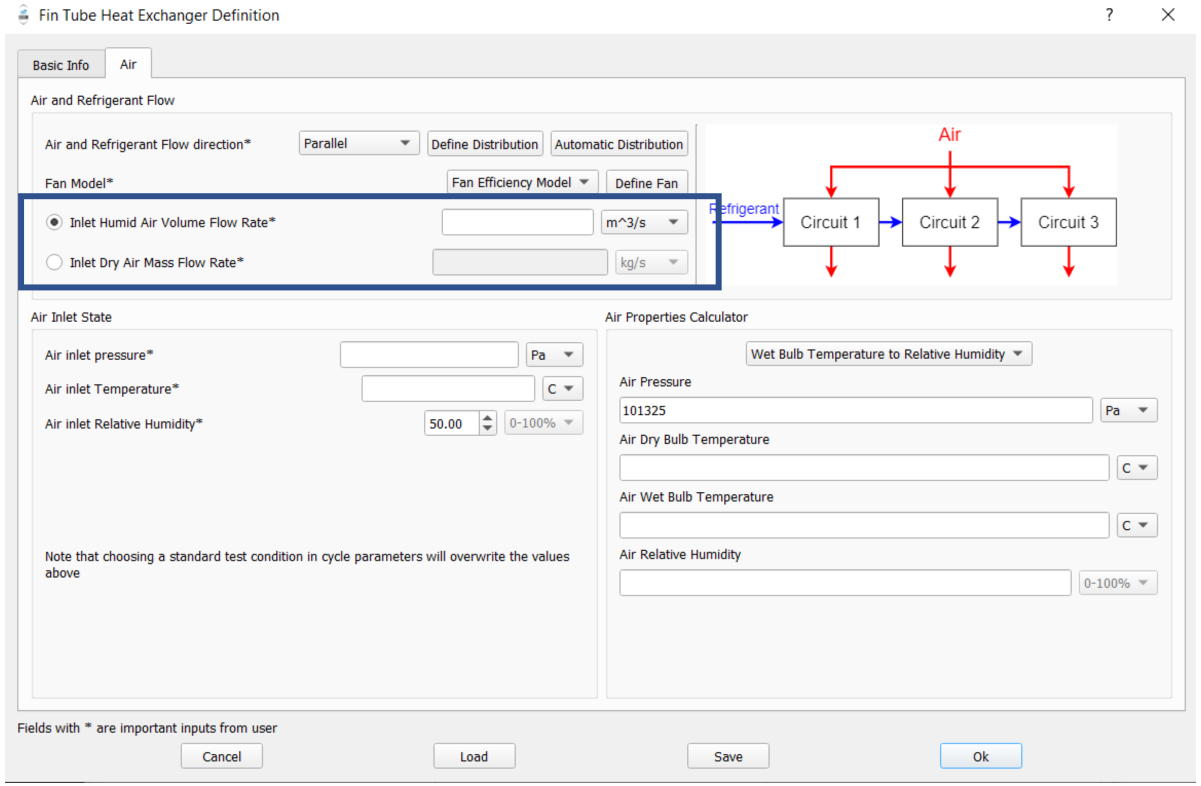
Fan Model: Fan Power Model

- In fan curve model, fan power is defined as a function of fan flowrate (in m^3/s)
- Fan power and heat exchanger air pressure drop will be used to calculate fan efficiency
- The user can define:
  - Any mathematical expression:
    -1722.9*x^2 + 451.29*x + 1.5714
  - a constant value: 40



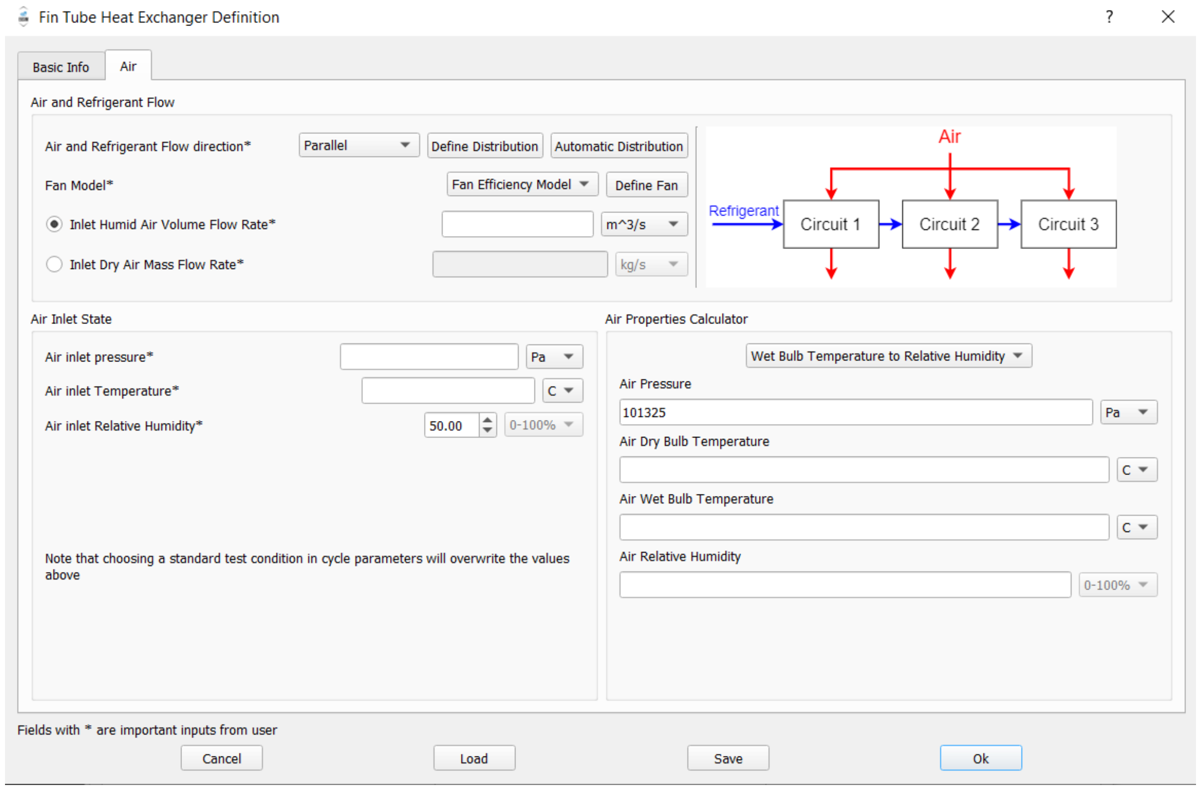
Air Flowrate

- The user can define either:
  - **Inlet Humid Air Volume Flow Rate**
  - **Inlet Dry Mass Flow Rate**
- If **Fan Curve Model** is selected, this value will be only used as an initial guess for the air flowrate



Air Inlet State

- The user can define the **Inlet Air State** to the heat exchanger using:
  - **Air Inlet Pressure**
  - **Air Inlet Temperature**
  - **Air Inlet Relative Humidity**
- This value will only be effective if **Custom** test condition is selected in section 2 (**Cycle Parameters**)
- An air properties calculator is available on the right to convert between wet-bulb temperature and relative humidity

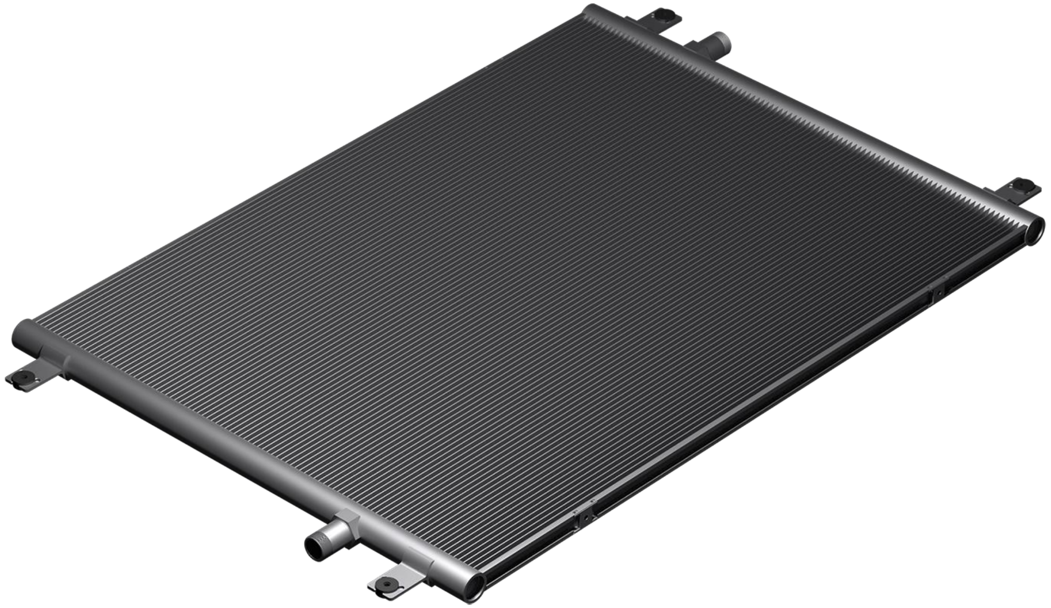


3.3 Microchannel heat exchangers

EGSim User Guide

Overview

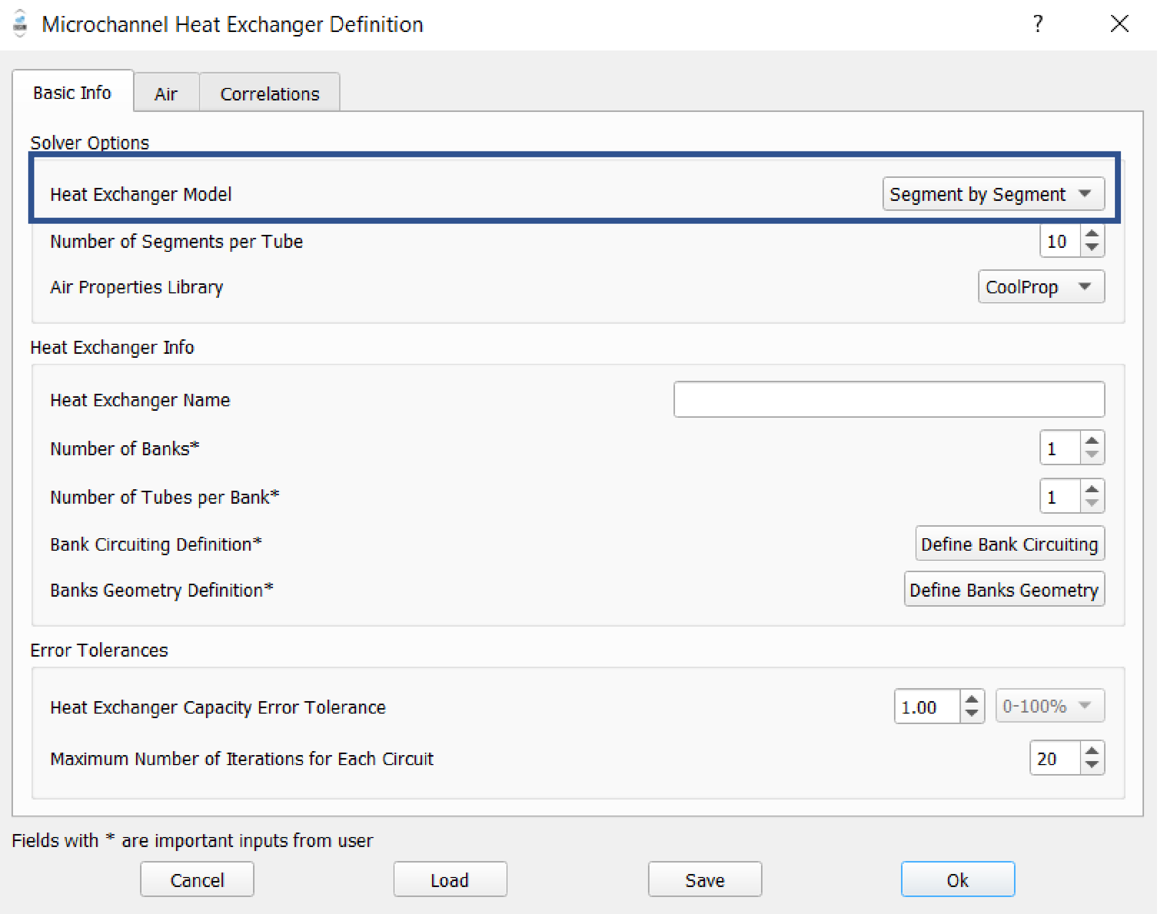
- The microchannel heat exchanger (MCHX) model is one of the two options available for heat exchangers in EGSim
- The MCHX can be used as a condenser or an evaporator (rarely)
- To model the MCHX, the user has to define:
  - The MCHX geometry
  - The air flowrate
  - The air inlet conditions



Basic Info Tab

Heat Exchanger Model

- The **Heat Exchanger Model** allows the user to define solver options
  - Phase-by-phase solver
  - Segment-by-segment solver

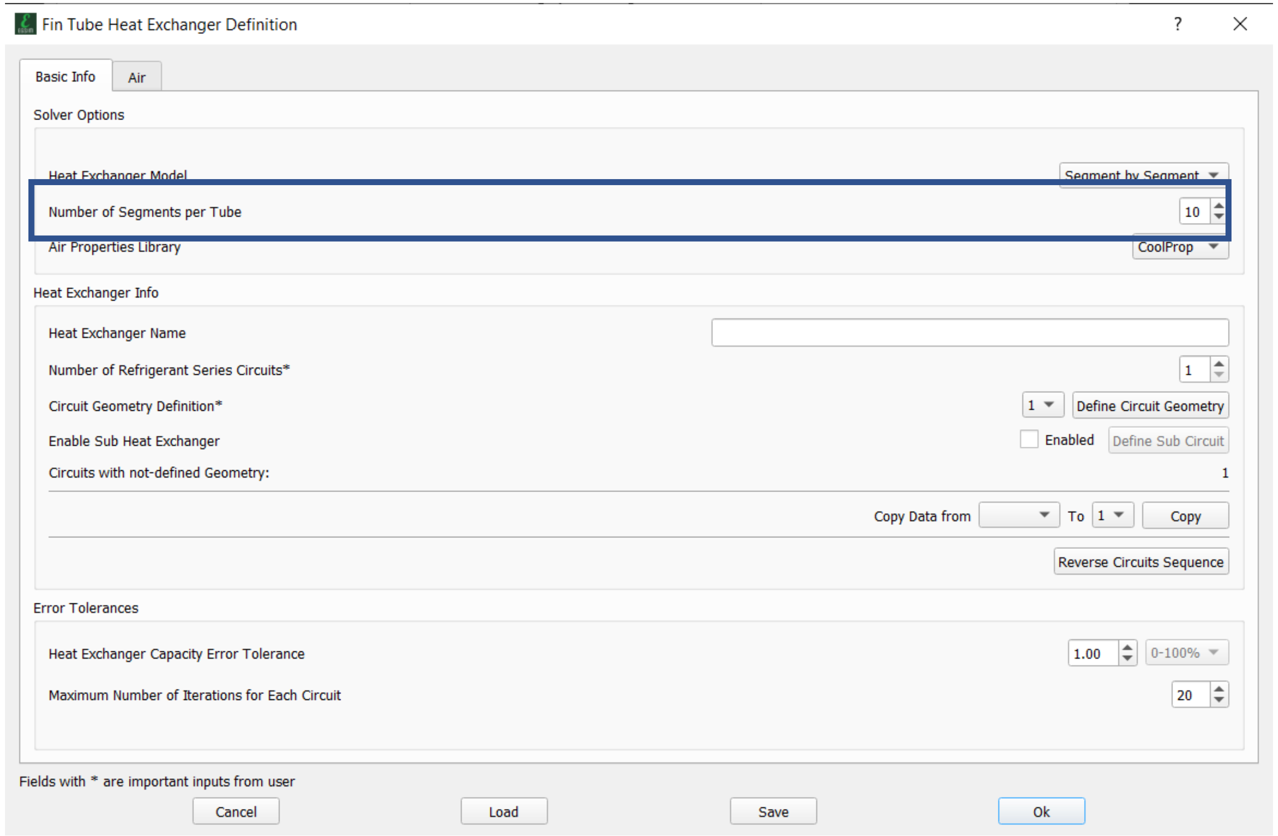


Phase-by-Phase Solver

- The heat exchanger is divided into a number of segments corresponding to the number of refrigerant phases
- The maximum number of segments is 3: superheated vapor, 2-phase, and subcooled liquid
- Advantages:
  - Fast
  - Acceptable accuracy
- Disadvantages:
  - Doesn’t account for tube circuit configuration
  - Doesn’t account for the effect of successive rows

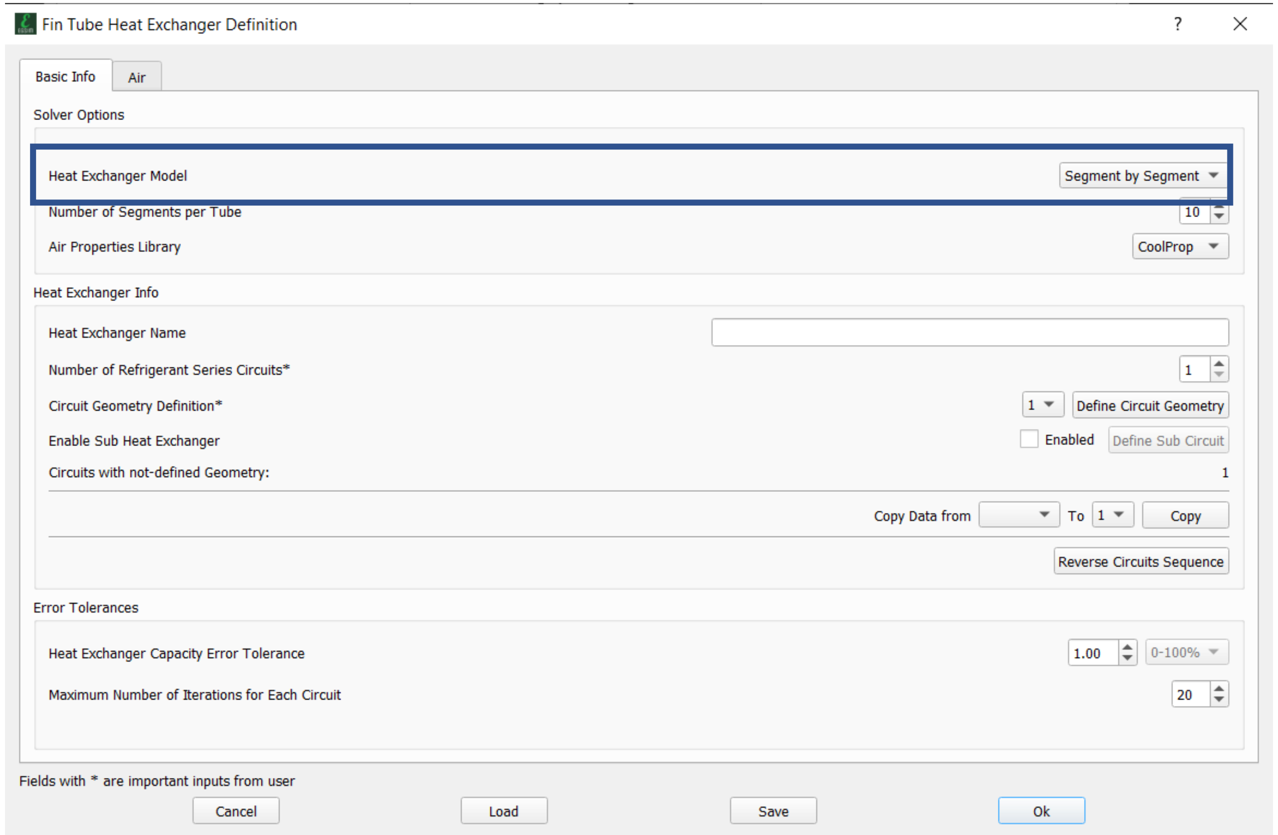
Segment-by-Segment Solver

- Each tube will be divided to **Number of Segments per Tube** defined by the user
- Each segment will be treated as a separate heat exchanger and solved



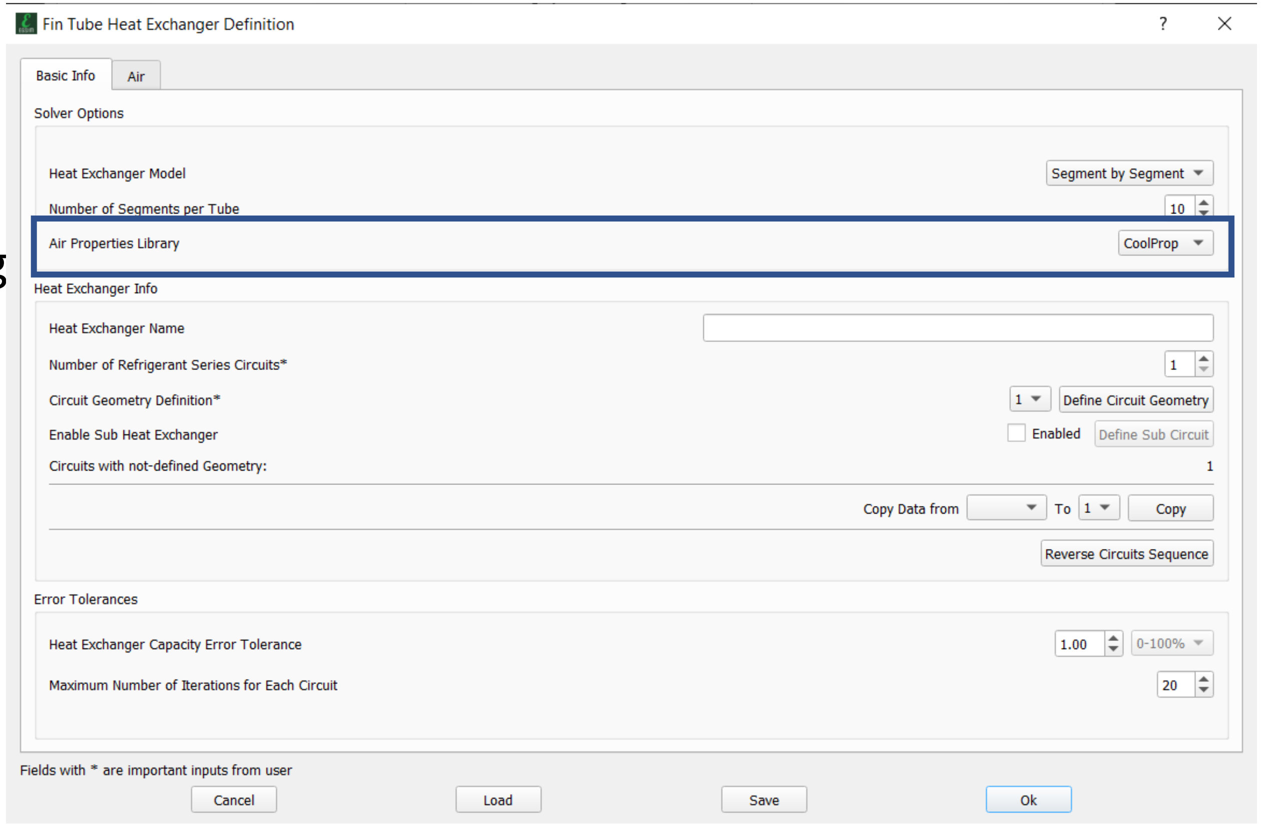
Segment-by-Segment Solver

- Advantages:
  - Highly accurate
  - Capable of evaluating the detailed tube circuit effects and successive tube rows
- Disadvantage: adding computational time, and hence slowing the overall simulation time



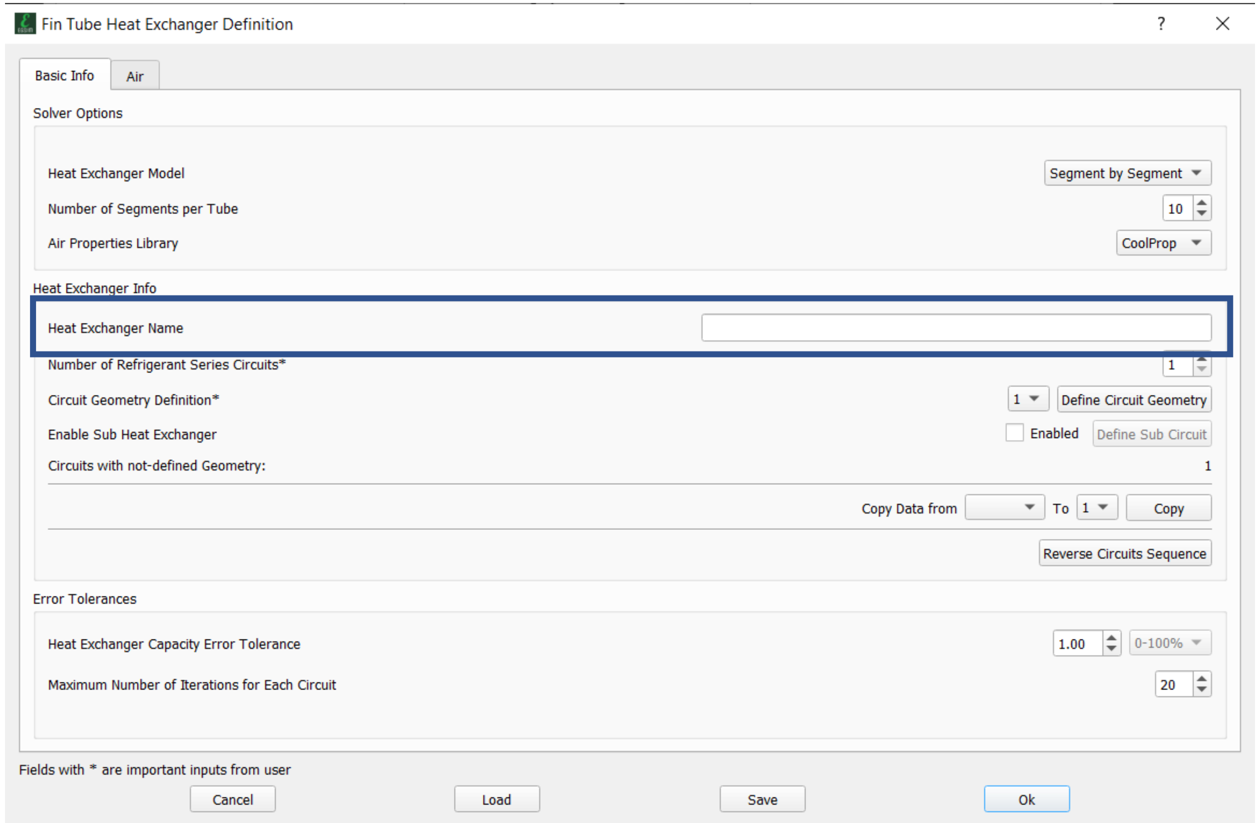
Air Properties Library

- The **Air Properties Library** defines the air properties calculations approach
  - **Coolprop** has the advantage of being accurate, reliable but relatively slow
  - **Psychrolob** uses ASHRAE approximations for air properties, which makes it faster to get air properties
- We advise the user to choose CoolProp



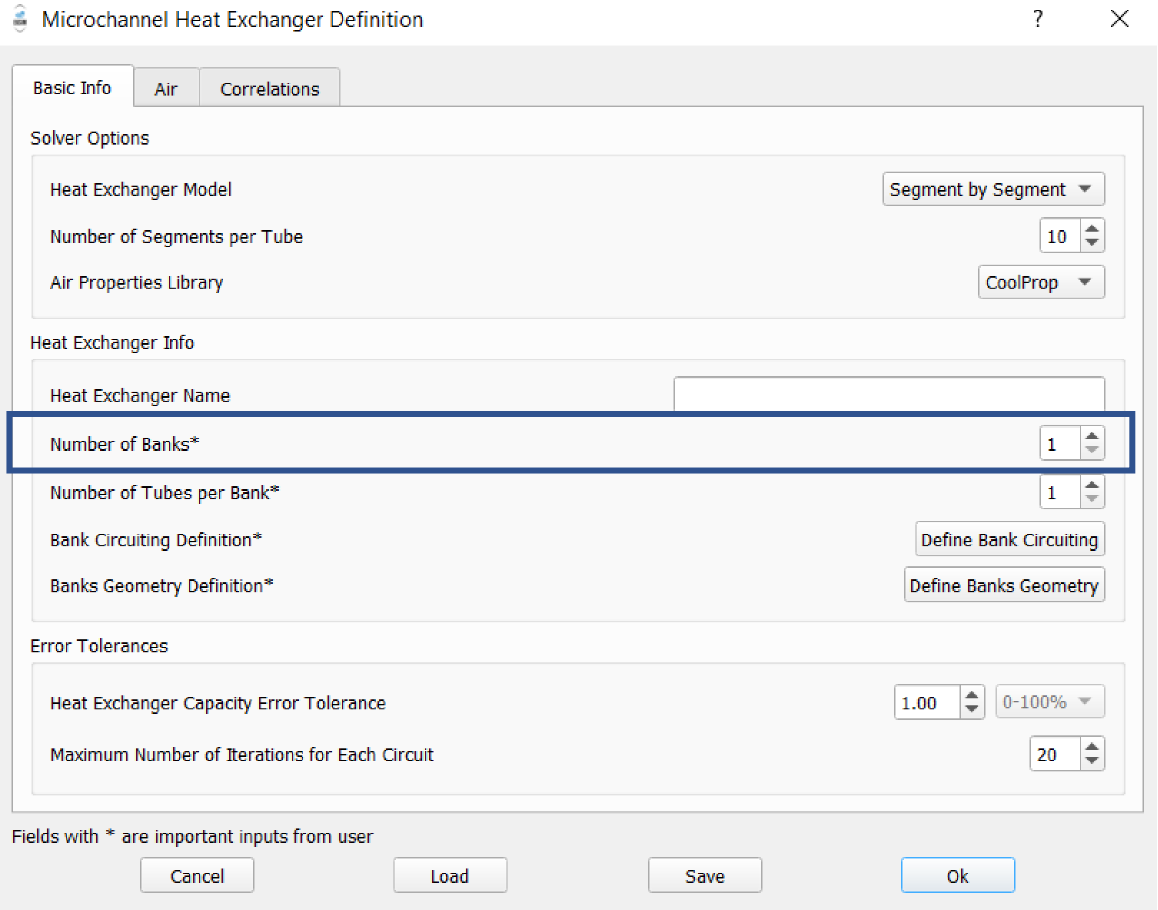
Heat Exchanger Name

- The **Heat Exchanger Name** is the name used to store the fin-tube heat exchanger into the database



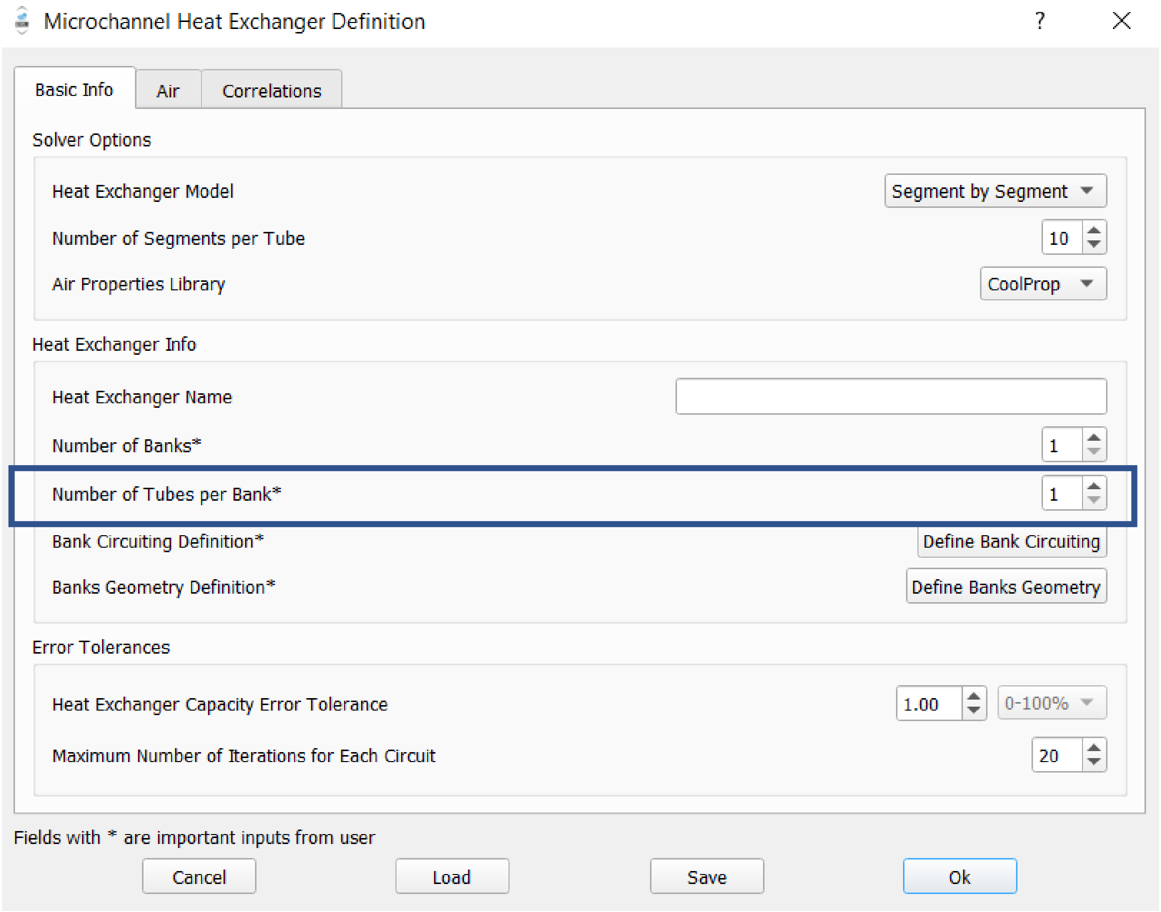
Number of banks (rows)

- The **Number of Banks** defines the number of tube banks or rows in the heat exchanger
- Typical MCHX are made of a single bank heat exchanger



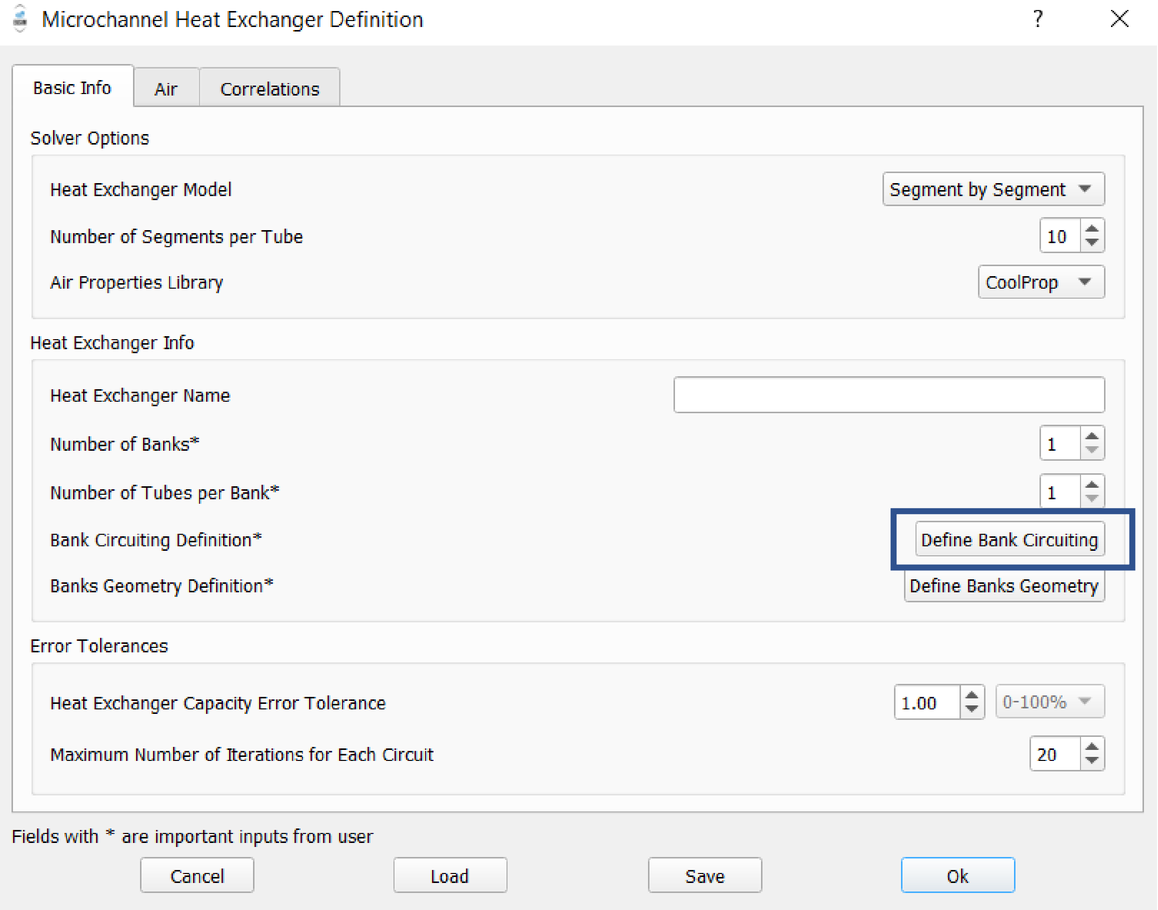
Number of Tubes per Bank (row)

- The **Number of Tubes per Bank** defines the total number of tubes per bank
- The total number of tubes for each bank should be the same across all banks



Bank Circuiting Definition

- The user can the tubes are circuited within each bank by clicking on **Define Bank Circuiting**



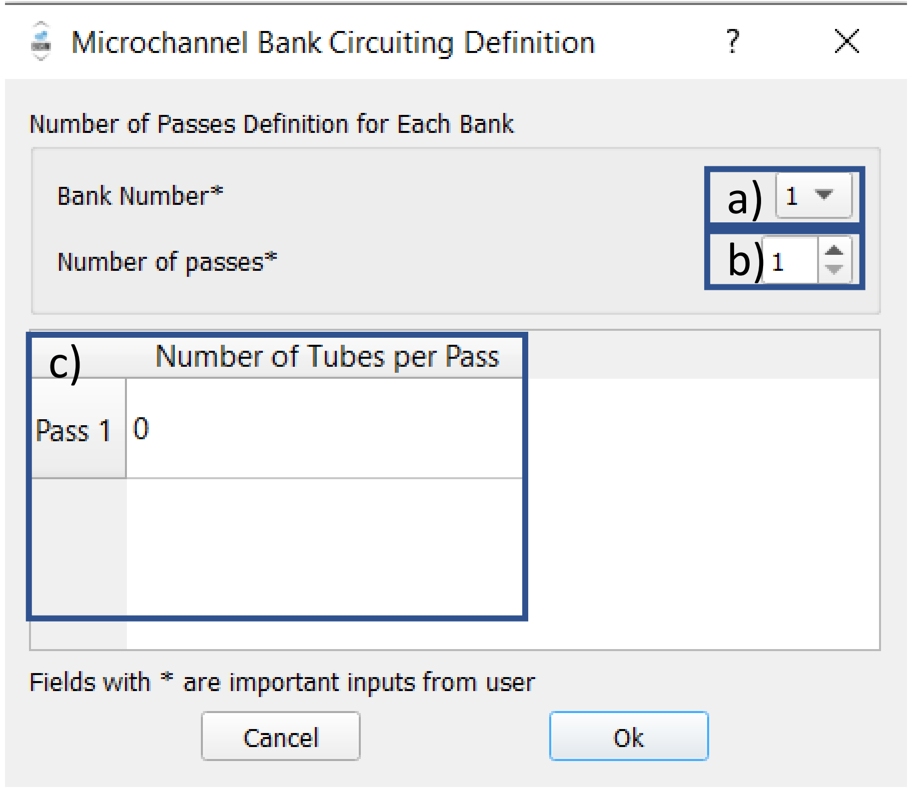
Bank circuiting definition, cont.

a) The user will have to first choose the bank (row) number to be defined

b) The user will then, choose the number of passes for that bank (row)

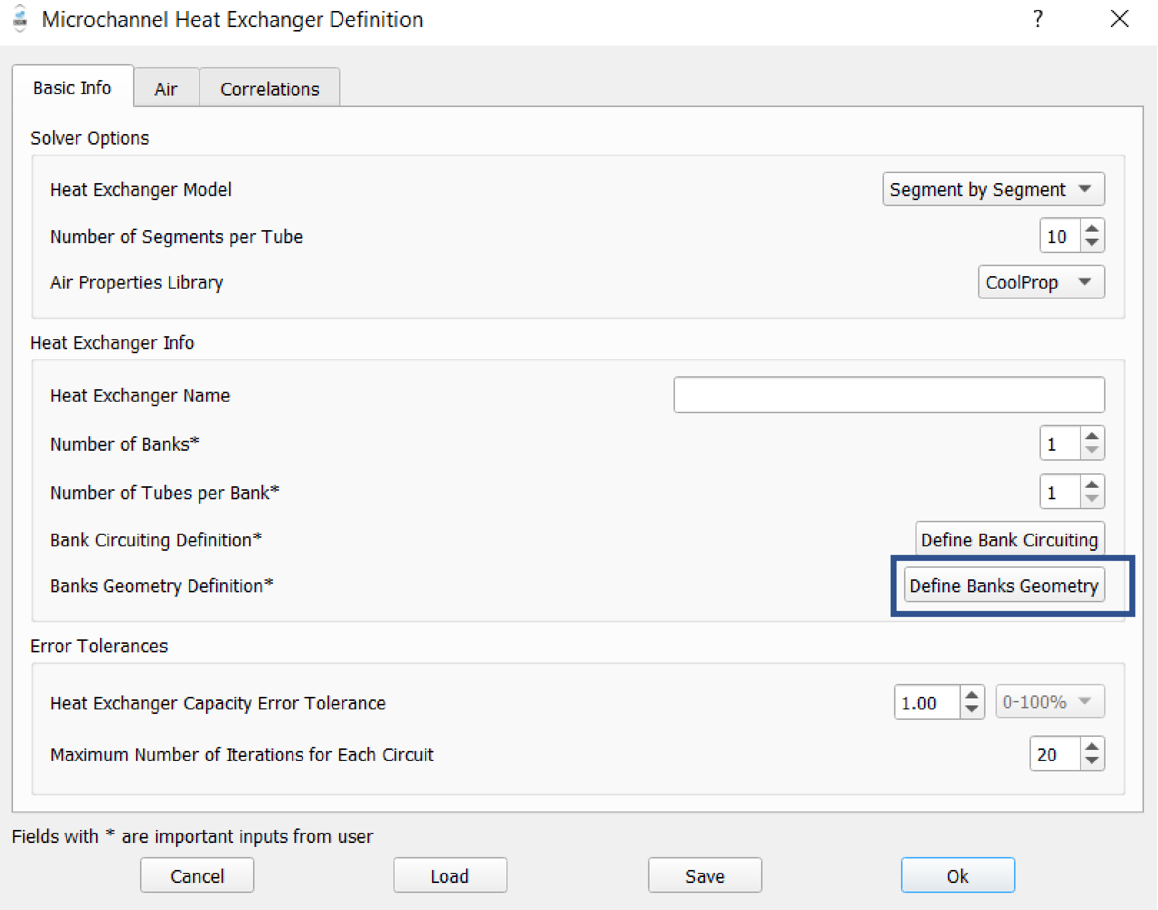
c) Finally, the user will need to define the number of tubes for each pass

Note: the summation of number of tubes for each pass must equal the total number of tubes per bank (row) defined before



Bank Geometry Definition

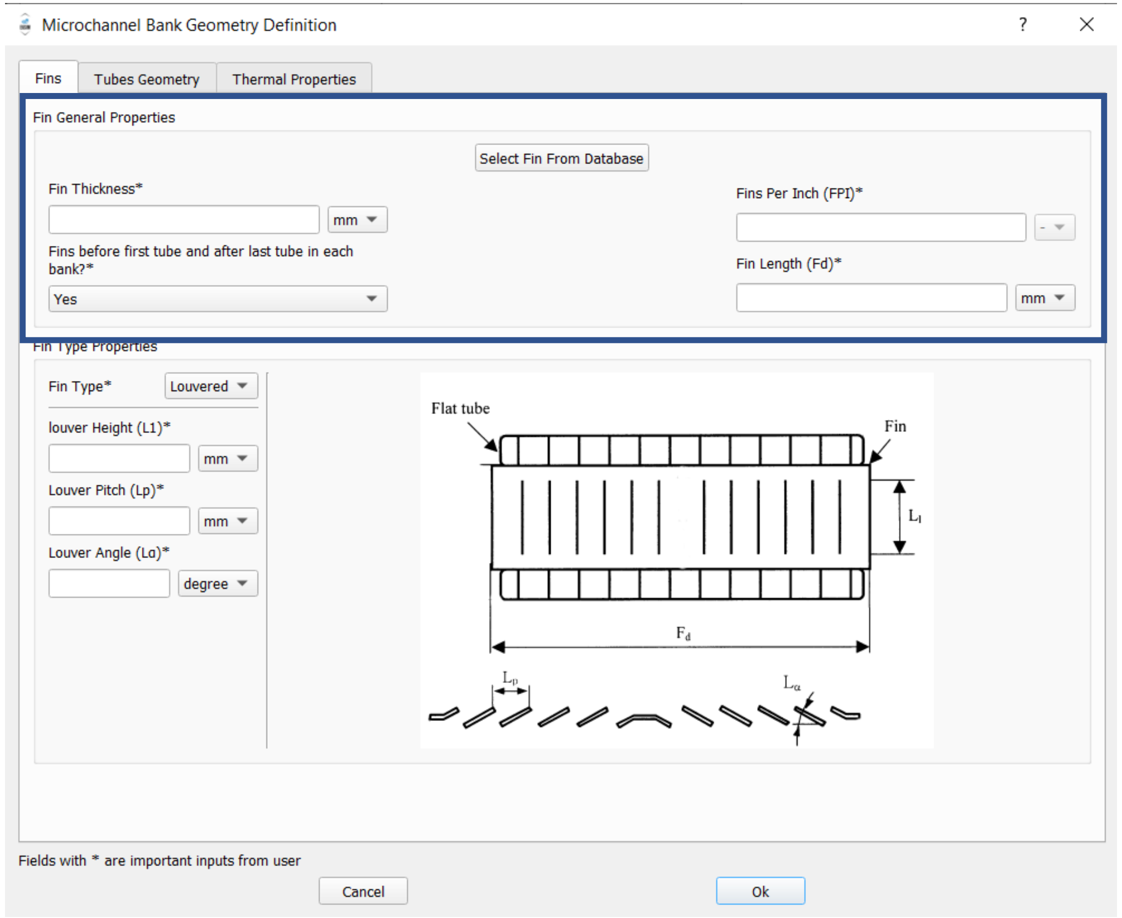
- The user can define the geometry of the banks (rows) by clicking on **Define Banks Geometry**



Microchannel Bank Geometry Definition

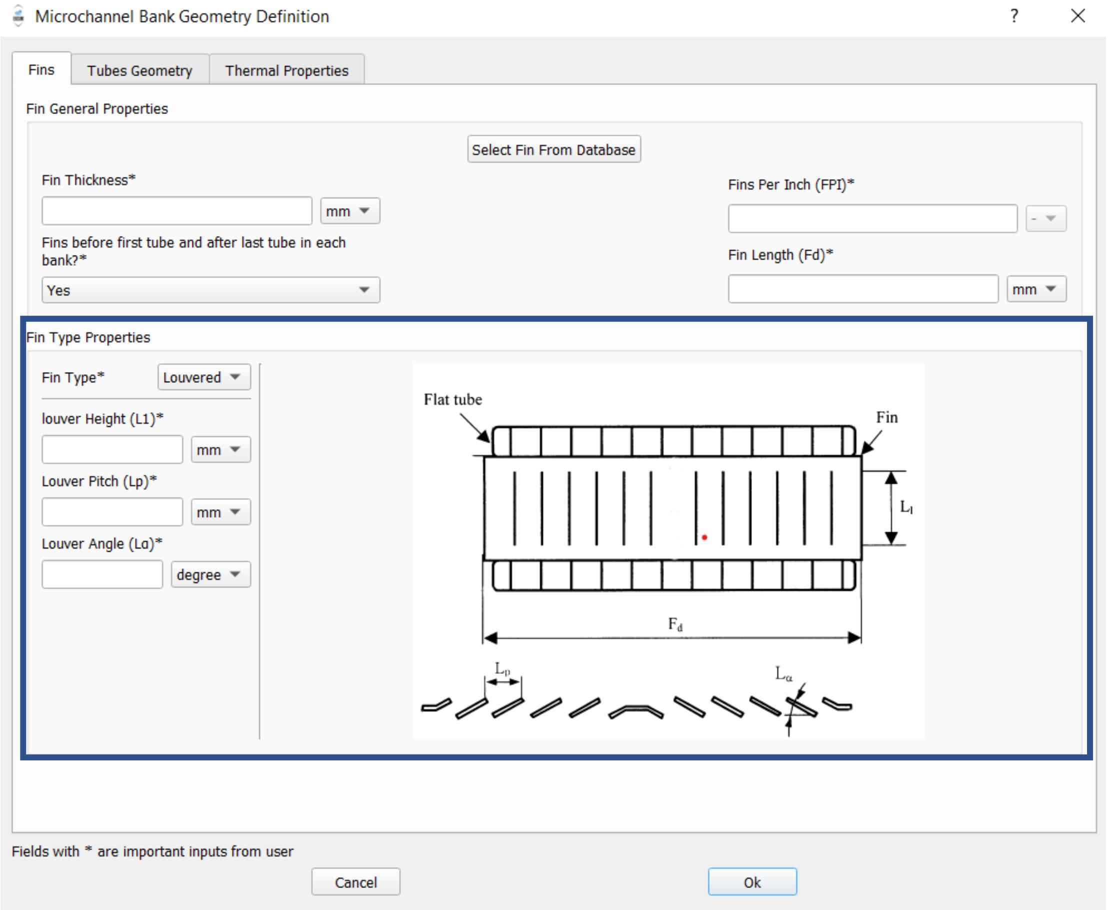
Fins

- The user can choose a predefined fin directly from **Select Fin from Database** button if any exist
- User-defined fins require the user to provide the values for **Fin Thickness**, **Fins Per Inch (FPI)** and **Fin Length (Fd)**
- If the modeled heat exchanger has more fin rows than tube rows, the user should select **Yes** for the dropdown list below “Fins before first tube and after last tube in each bank?”



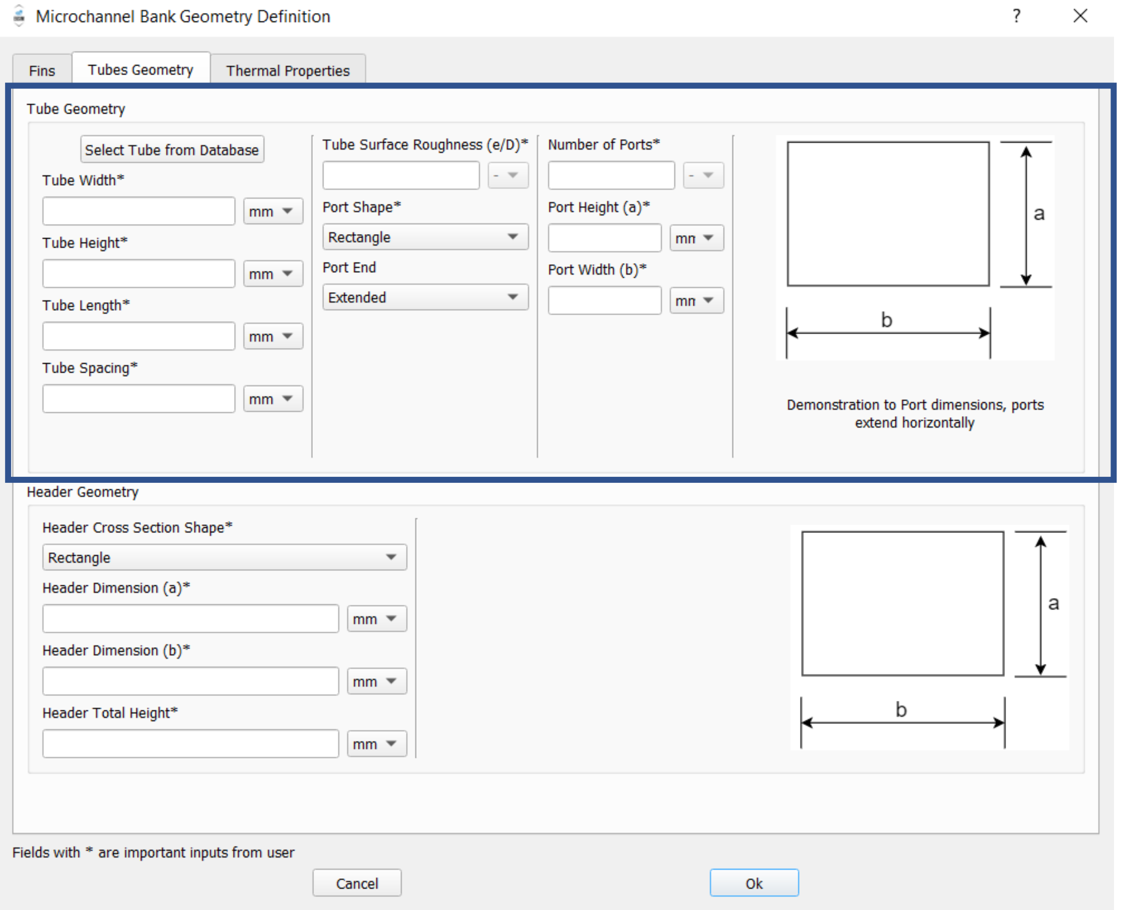
Fins, cont.

- Choose a Fin Type from the available list, currently EGSim only supports Louvered Fins
- For each fin type, additional parameters of the fin geometry might be needed, as defined in the illustration shown on the right



Tubes Geometry

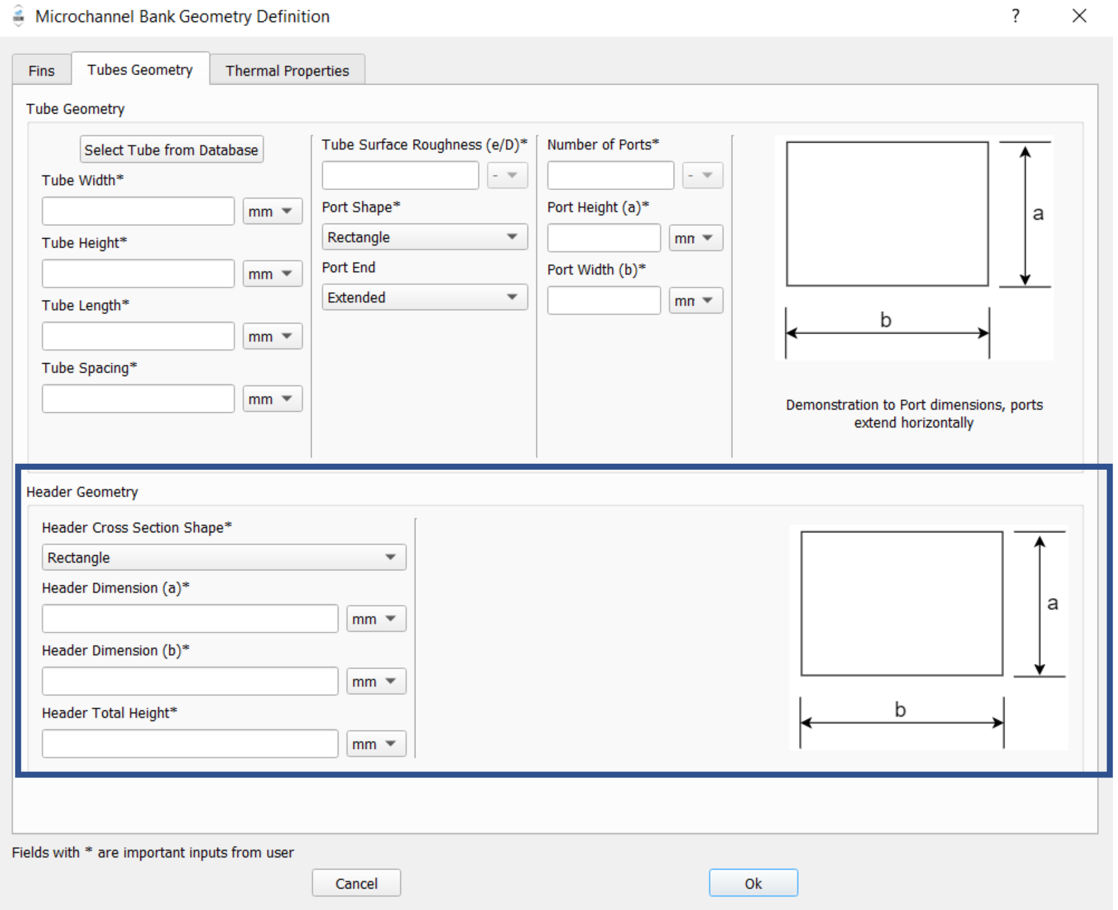
- The user can choose a predefined tube directly from **Select Tube from Database** button if any exist
- User-defined tubes require the user to provide detailed geometry including:
  - **Tube Width, Height, Length, and Spacing**
  - **Port Shape** (Rectangular, Circular, or Triangular)
  - **Port End** (Normal, or Extended)
  - Port Geometry (**Number of Ports, Port Height, and Port Width**)



Header Geometry

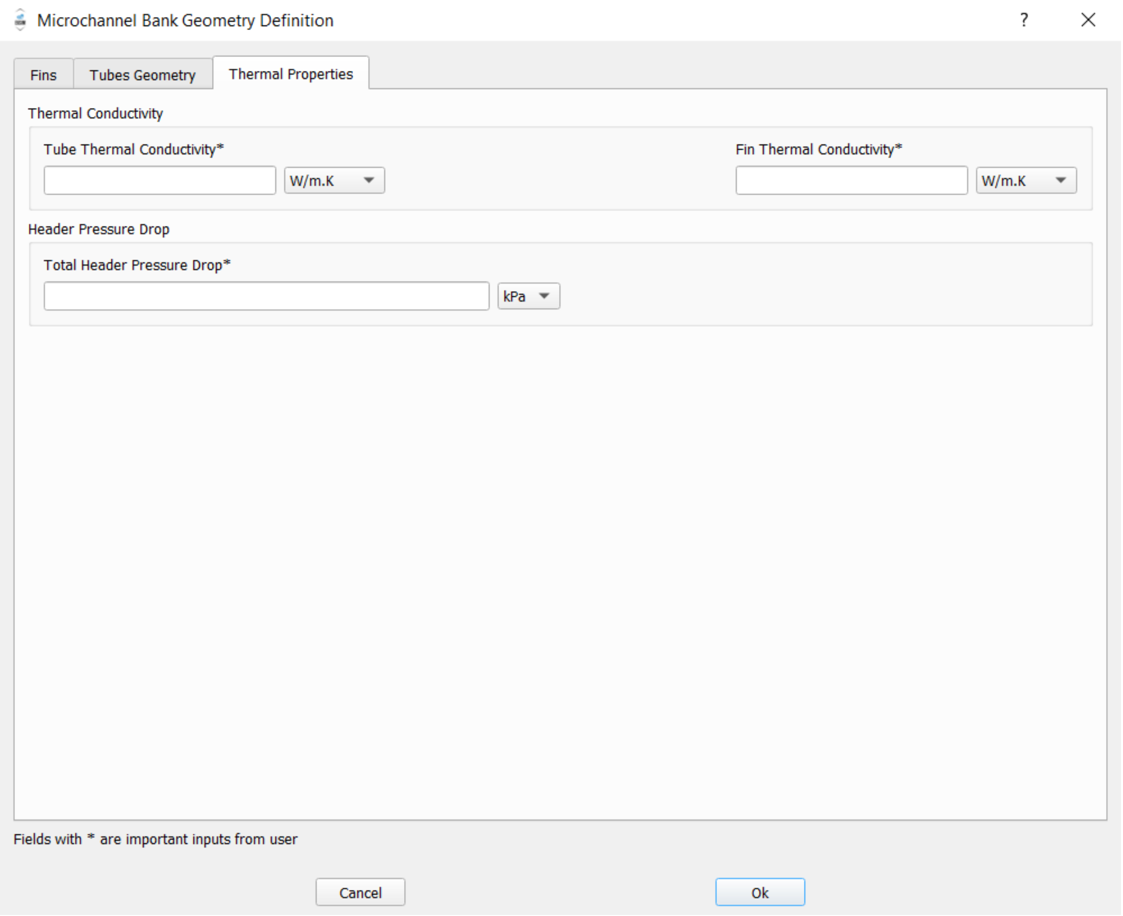
- Header geometry will affect the refrigerant charge calculation
- The user has to specify the **Header Cross Section Shape**:
  - Rectangular
  - Circular
- The user must define the header dimensions including the **Header Dimension (a) and (b) and Header Total Height**

Note: the Header total height must be larger than total height of tubes + fin rows



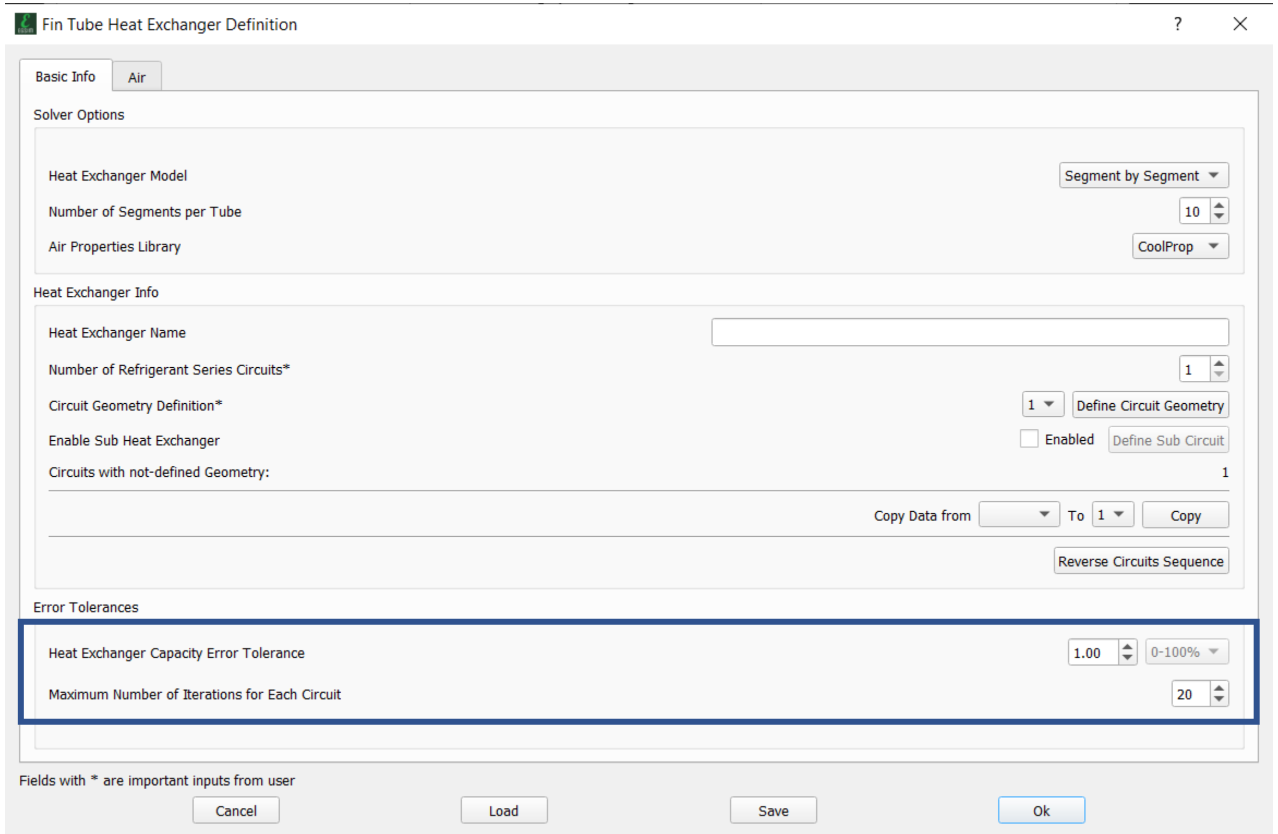
Thermal Properties

- The user must provide the **Tube Thermal Conductivity** and the **Fin Thermal Conductivity** these values are used for the overall heat transfer calculations
- The user must also define the **Total Header Pressure Drop** - ; This includes the pressure drop in the headers only, and not the tubes



Error Tolerances

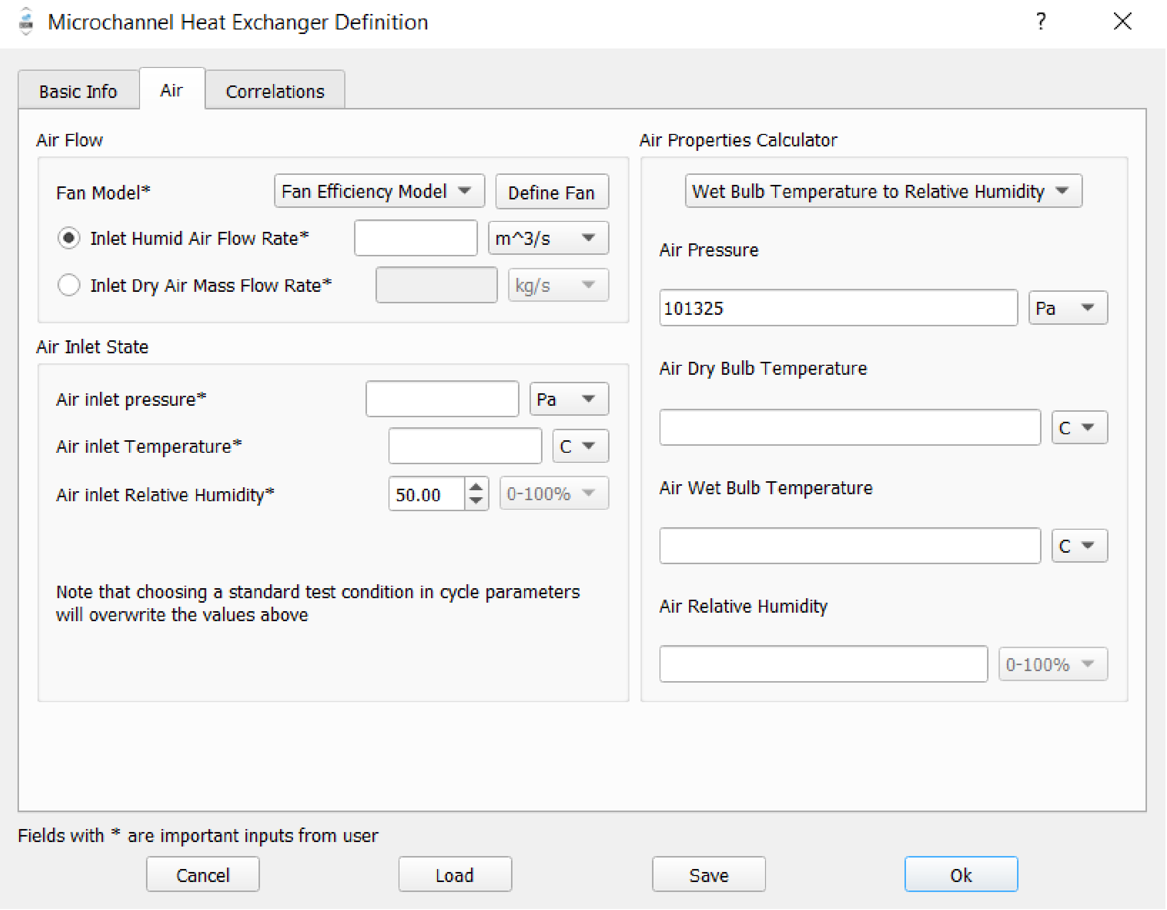
- Segment-by-Segment solver relies on trial-and-error, the solver convergence criteria is the difference in heat exchanger capacity
- The **Heat Exchanger Capacity Error Tolerance** defines the maximum allowed error
- The **Maximum Number of Iterations for Each Circuit** is used to limit the solver time, the user can select how many iterations can be reached before the solver exits even if the convergence criteria are not met



Air Tab

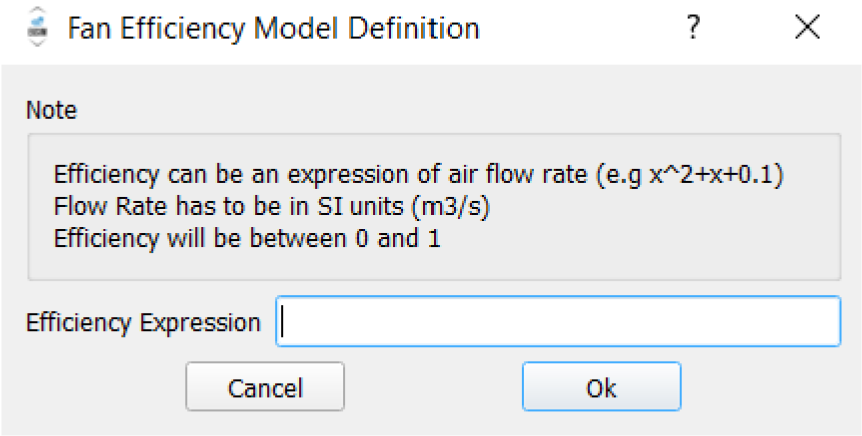
Fan Model

- The user can choose from 3 available fan models in EGSim:
  - **Fan efficiency model**
  - **Fan curve model**
  - **Fan Power model**



Fan Model: Fan Efficiency Model

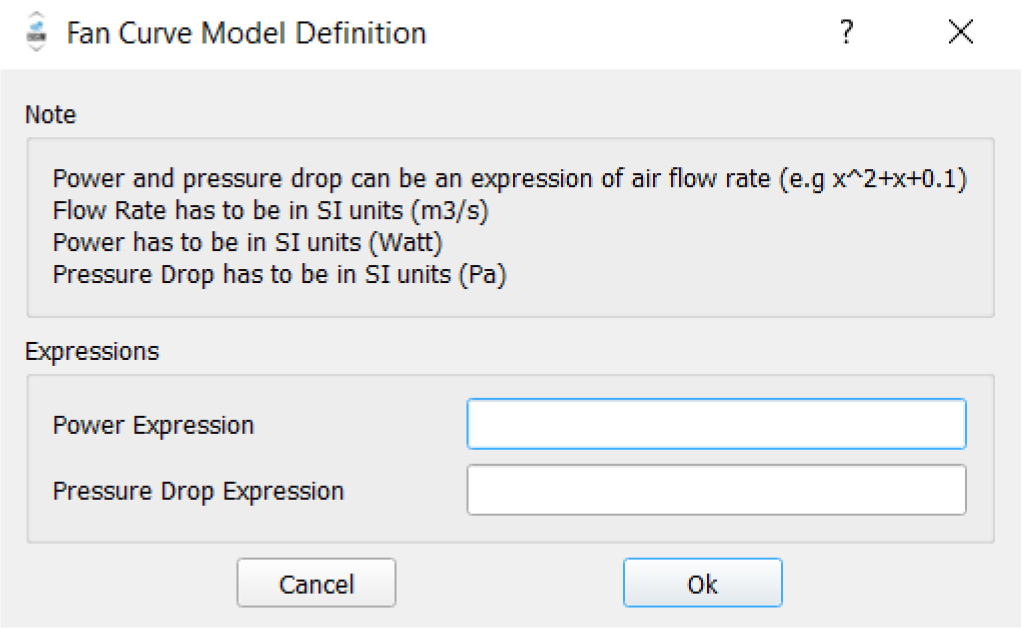
- In fan efficiency model, fan efficiency is defined as a function of the fan flowrate (in m^3/s)
- From fan efficiency and heat-exchanger air pressure drop, fan power will be calculated
- The user can define:
  - Any mathematical expression:
    -28.292*x^2 + 7.6344*x - 0.0183
  - A constant value: 0.6



Fan Model: Fan Curve Model

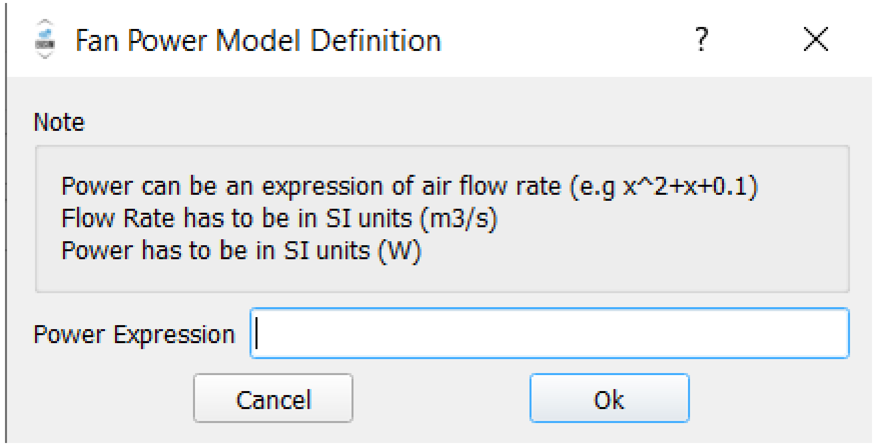
- In fan curve model, fan power and fan pressure-rise are defined as a function of fan flowrate (in m^3/s)
- Fan curve will be solved with heat-exchanger system curve to find operating air flowrate and corresponding fan power and efficiency
- The user can define:
  - Any mathematical expression:

    -1722.9*x^2 + 451.29*x + 1.5714
  - A constant value: 40



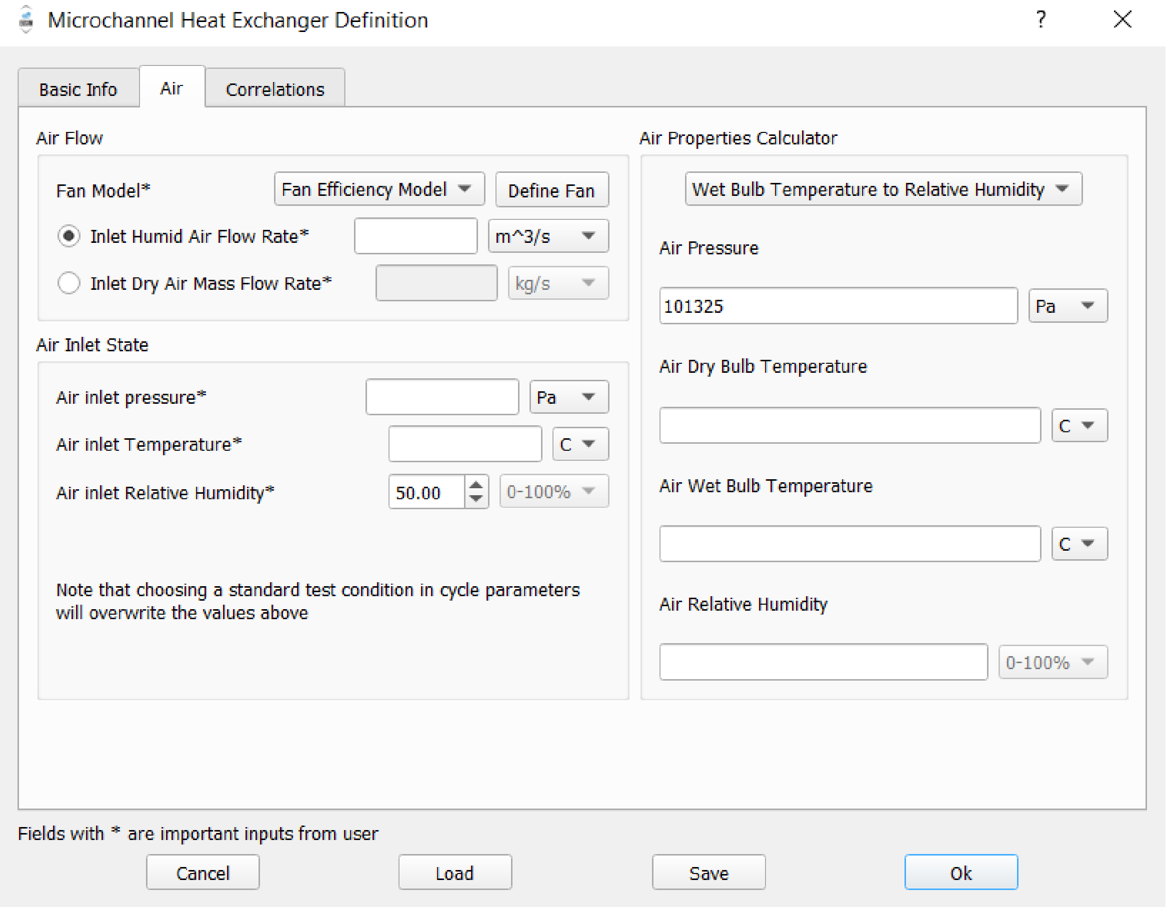
Fan Model: Fan Power Model

- In fan curve model, fan power is defined as a function of fan flowrate (in m^3/s)
- Fan power and heat exchanger air pressure drop will be used to calculate fan efficiency
- The user can define:
  - Any mathematical expression:
    -1722.9*x^2 + 451.29*x + 1.5714
  - a constant value: 40



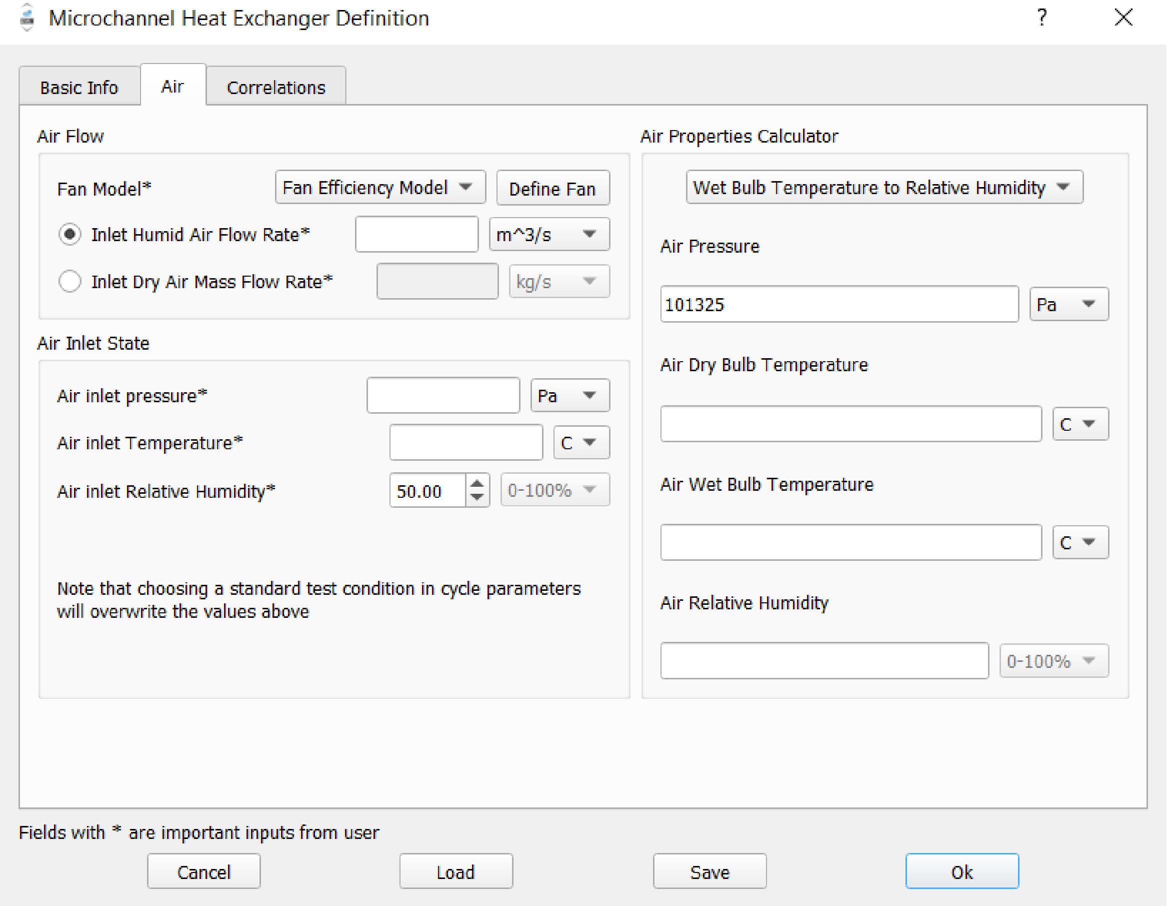
Air flowrate

- The user can define either:
  - **Inlet Humid Air Volume Flow Rate**
  - **Inlet Dry Mass Flow Rate**
- If **Fan Curve Model** is selected, this value will be only used as an initial guess for the air flowrate



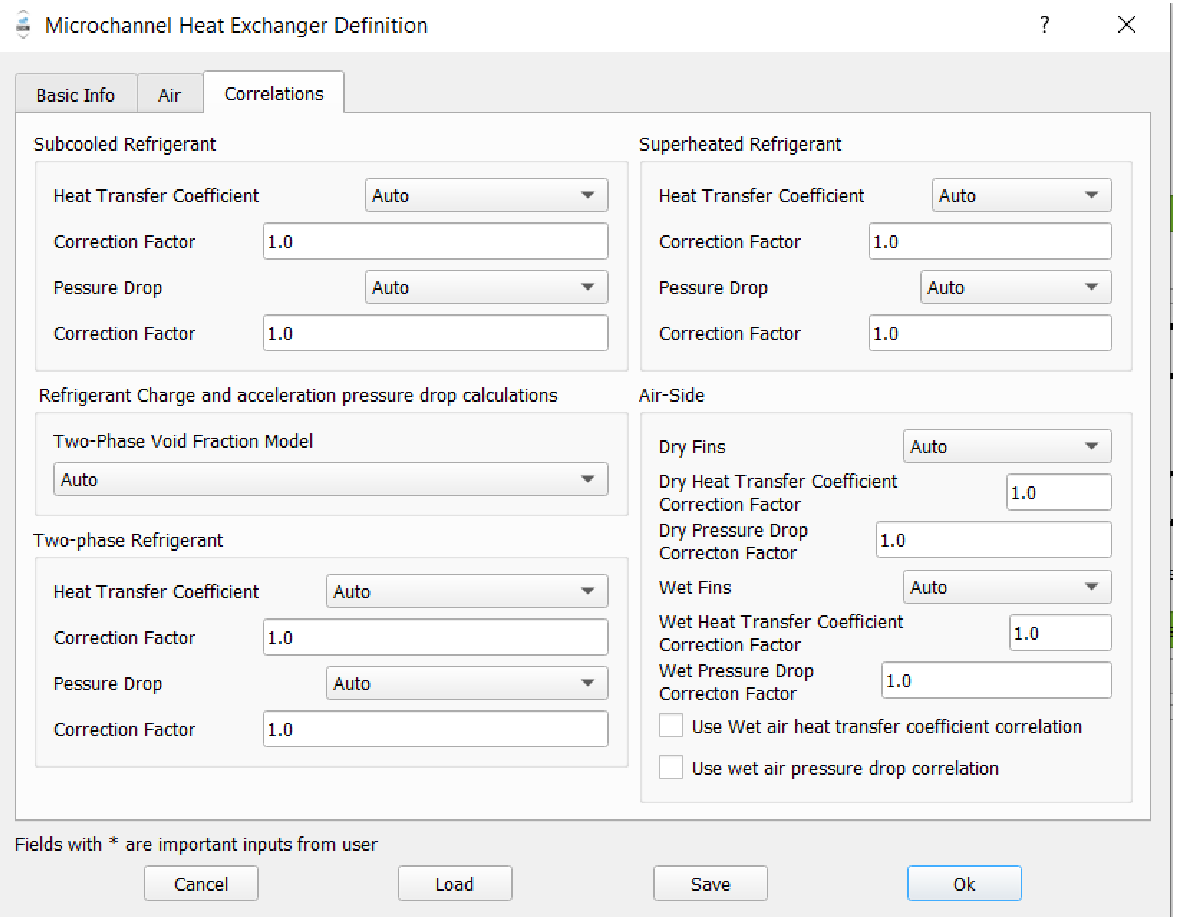
Air Inlet state

- The user can define the **Inlet Air State** to the heat exchanger using:
  - **Air Inlet Pressure**
  - **Air Inlet Temperature**
  - **Air Inlet Relative Humidity**
- This value will only be effective if **Custom** test condition is selected in section 2 (**Cycle Parameters**)
- An air properties calculator is available on the right to convert between wet-bulb temperature and relative humidity



Correlations

- On this tab, the user can define correlations to use for the evaluation of heat transfer and pressure drop
  - Refrigerant side **Subcooled**, **Two-phase**, and **Superheated**
  - Air-side; dry and wet
- Selecting “**Auto**” will allow the software to choose the most suitable correlation according to a smart algorithm which takes the geometric and physical conditions of the case into account
- **Correction Factors** can be used to provide needed correction to the heat transfer and pressure drop evaluation for validation purposes

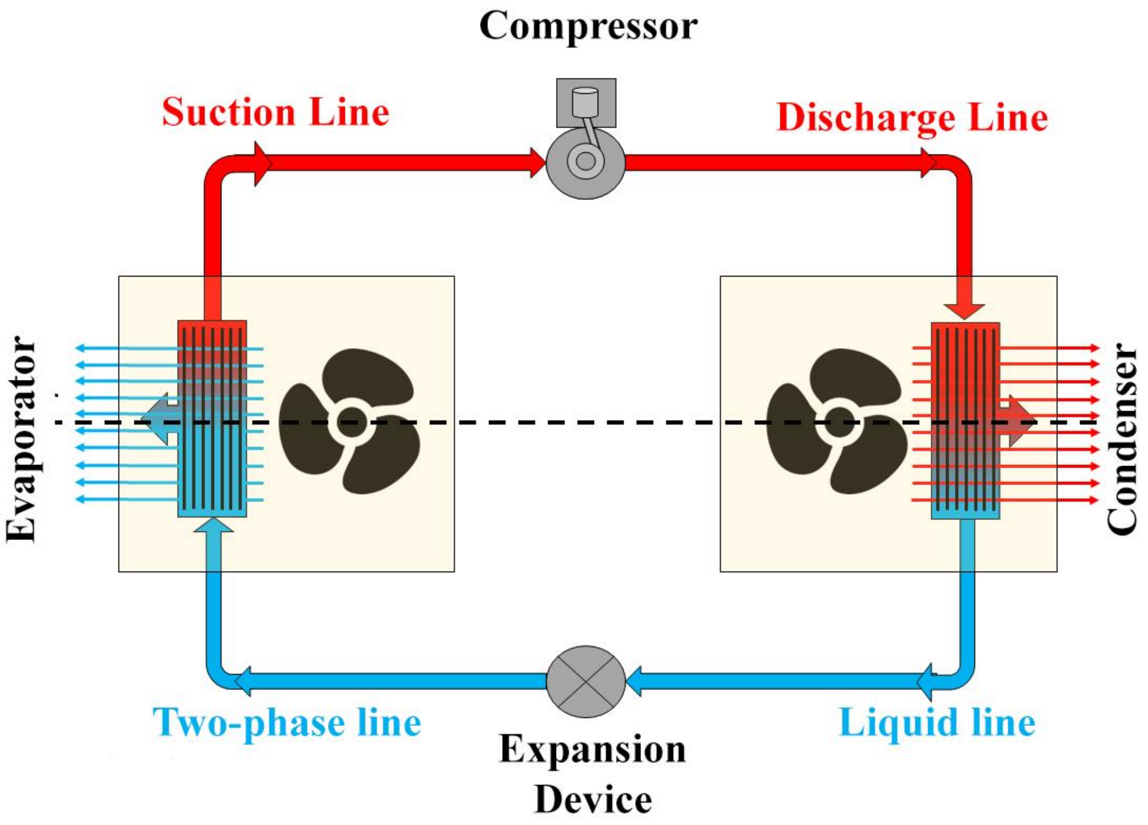


3.4 Connecting lines

EGSim User Guide

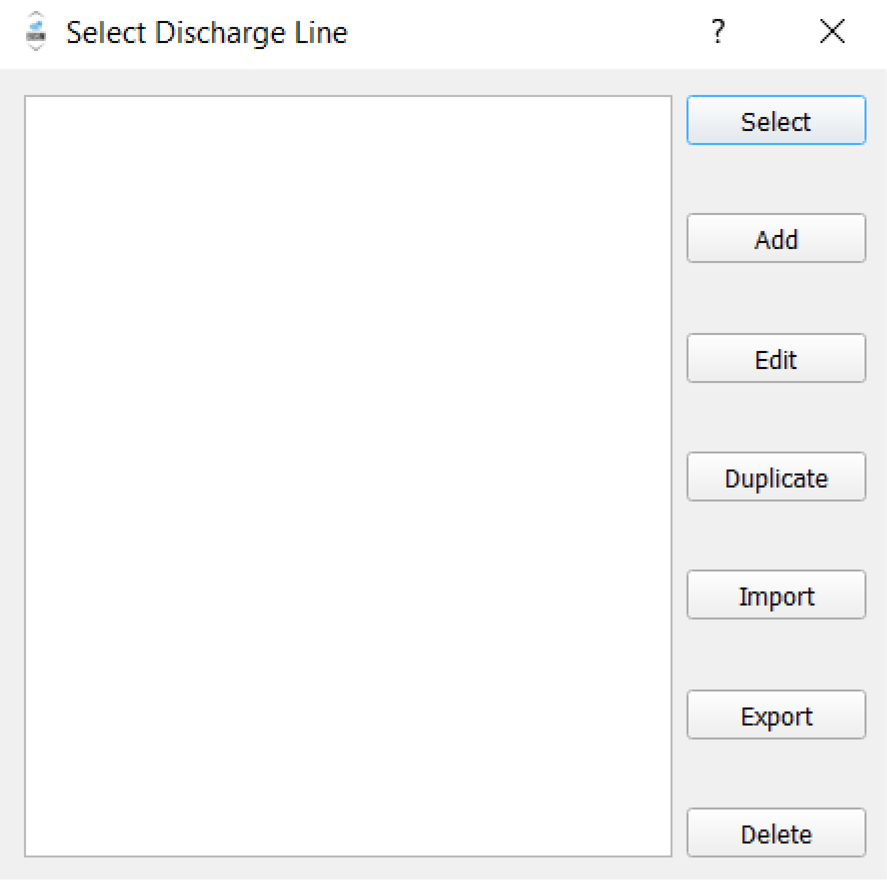
Overview

- Connecting lines are used to connect the 4 components of the cycle
  - Liquid line: connecting the condenser to the expansion device
  - Two-phase line: connecting the expansion device to the evaporator
  - Suction line: connecting the evaporator to the compressor
  - Discharge line: connecting the compressor to the condenser
- If the cycle is switched between heating and cooling modes, the lines will be automatically swapped



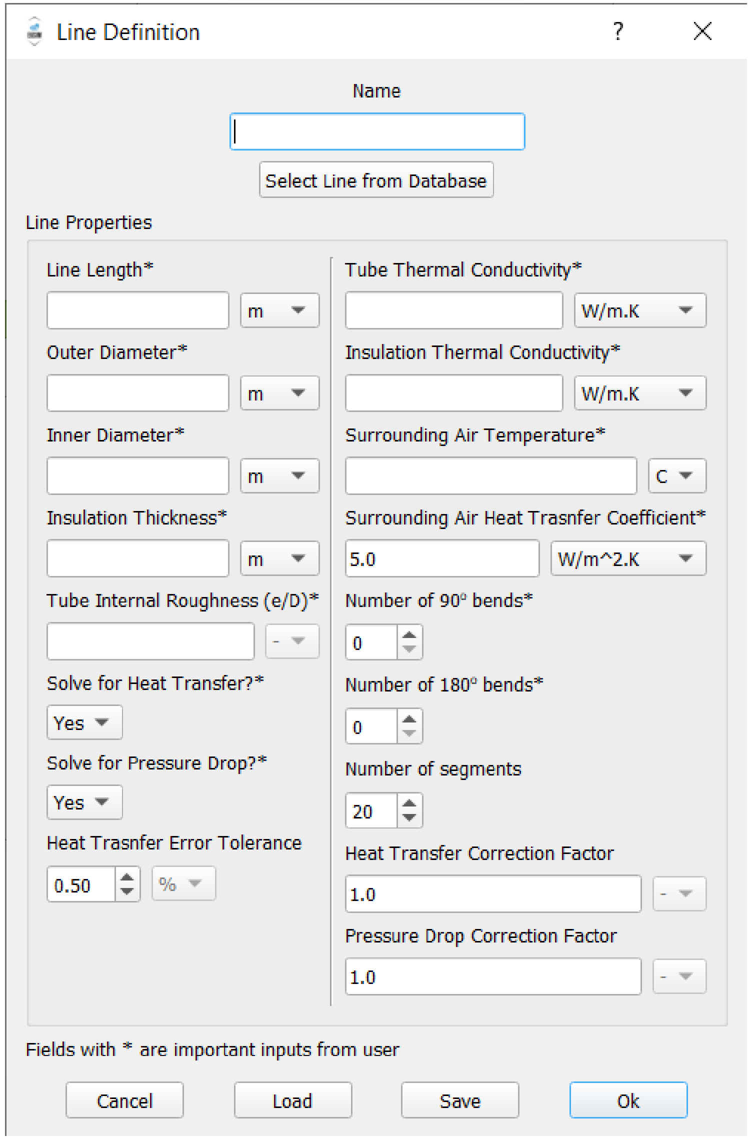
Connecting lines database

- The **Connecting Lines** database are shared between the 4 types of lines
- To add a new connecting line, click **Add**



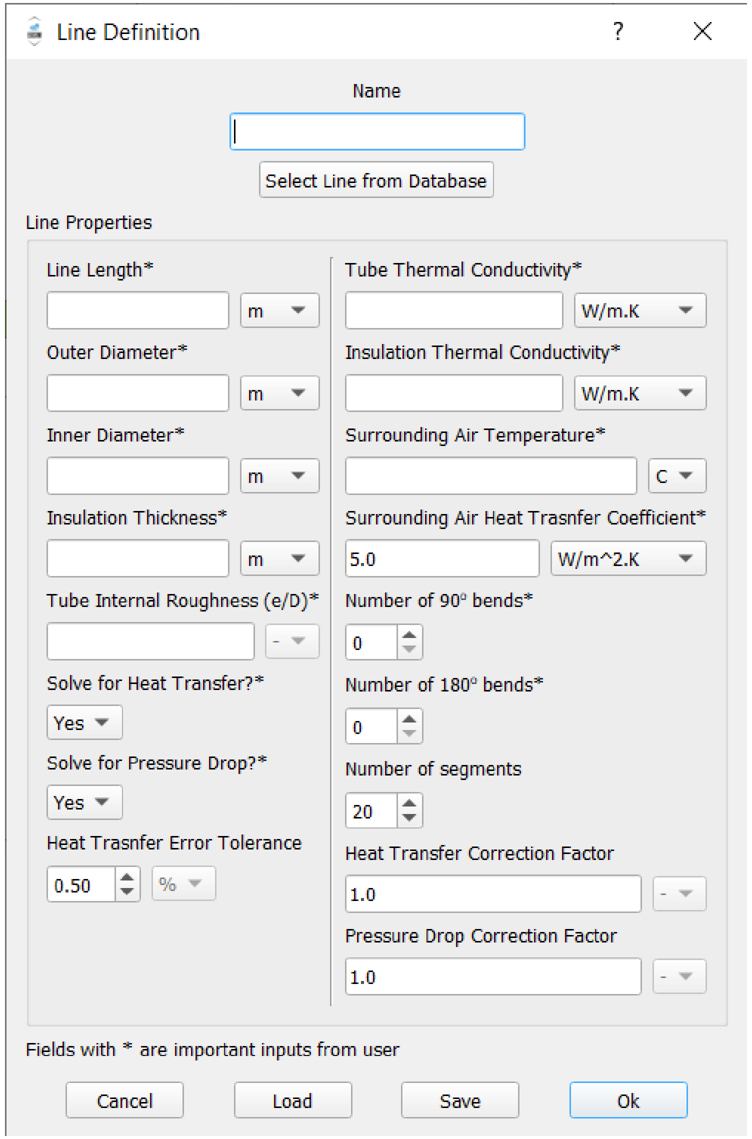
Line Definition

- The user must create a name for the line that will appear in Connecting Lines database
- The user can select a line from the database, which will define most of the geometric features of the line
- Otherwise, the user can enter all the geometric properties of the line manually



Line Definition, cont.

- If line consists of a uniform outer and inner diameters, then enter the **Line Length**, **Outer Diameter** and **Inner Diameter**
- If the line consists of multiple sections each having different outer and inner diameters, then the user should enter the outer and inner diameters corresponding to the longest section

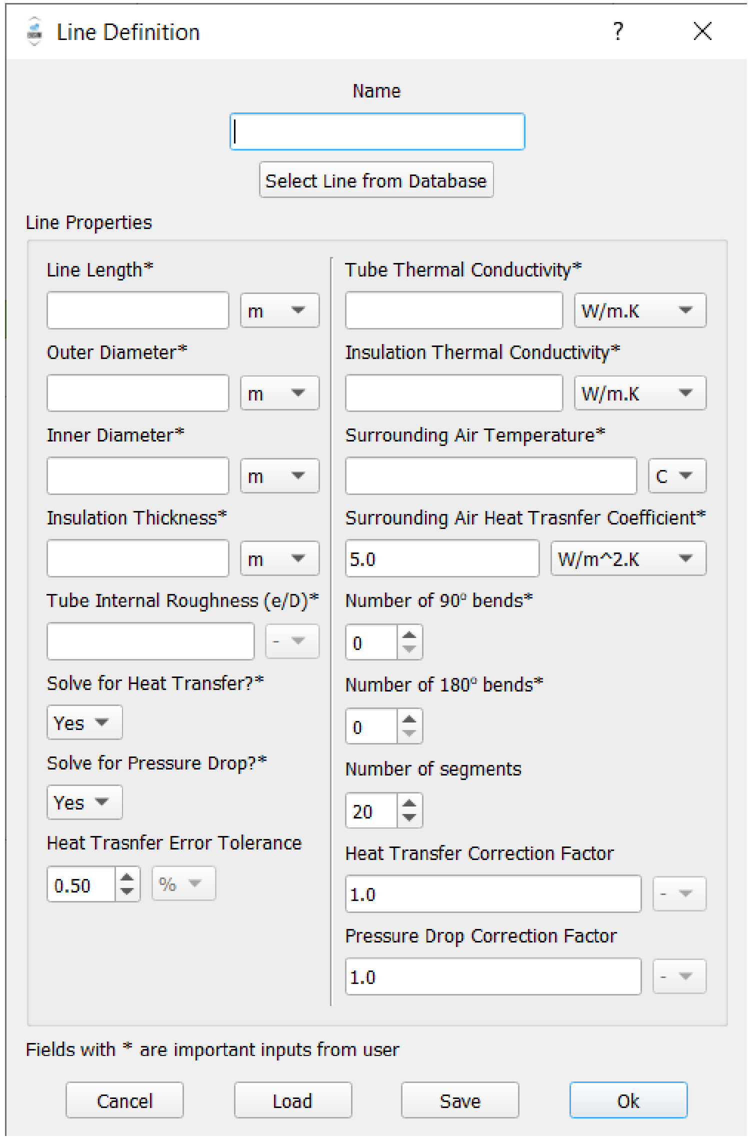


Line Definition, cont.

- The line length then will be an effective line length calculated using the following formula:

$$L_{eff} = \sum L_{section} * \left(\frac{D_{section}}{D_{chosen}}\right)^2$$

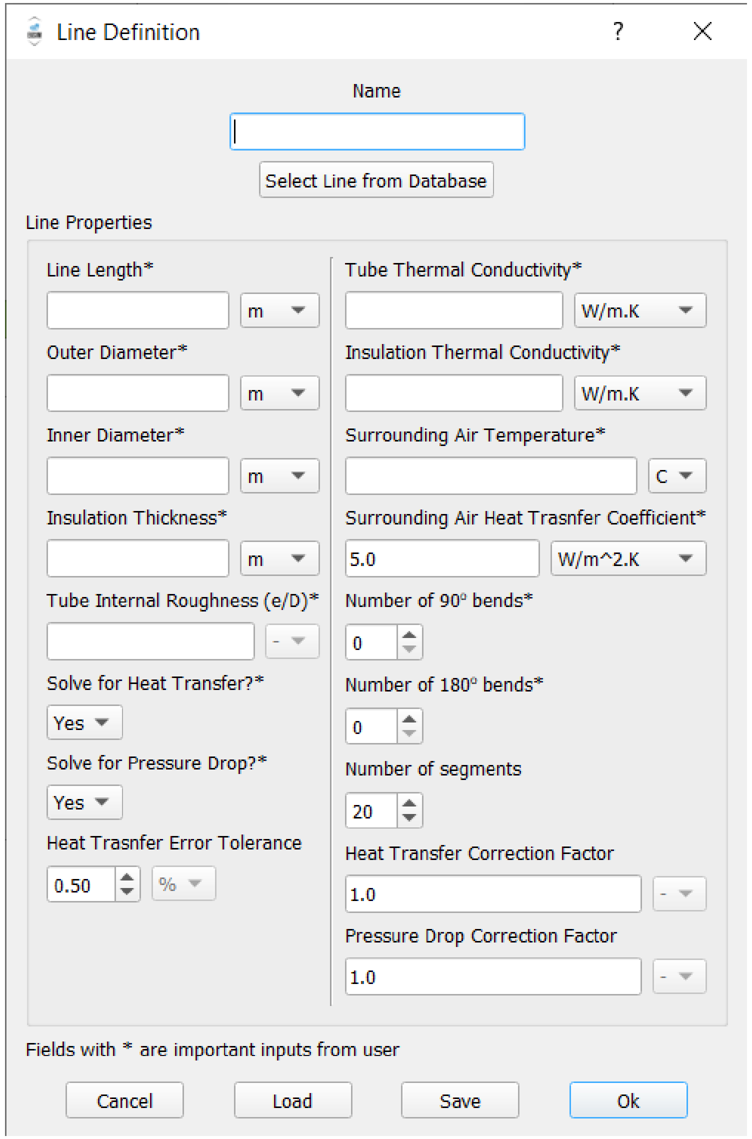
- Where:
  - L_{eff} is the effective line length
  - L_{part} is the length of the section
  - $D_{section}$ is the inner diameter of the section
  - D_{chosen} is the inner diameter of the largest section



Line Definition, cont.

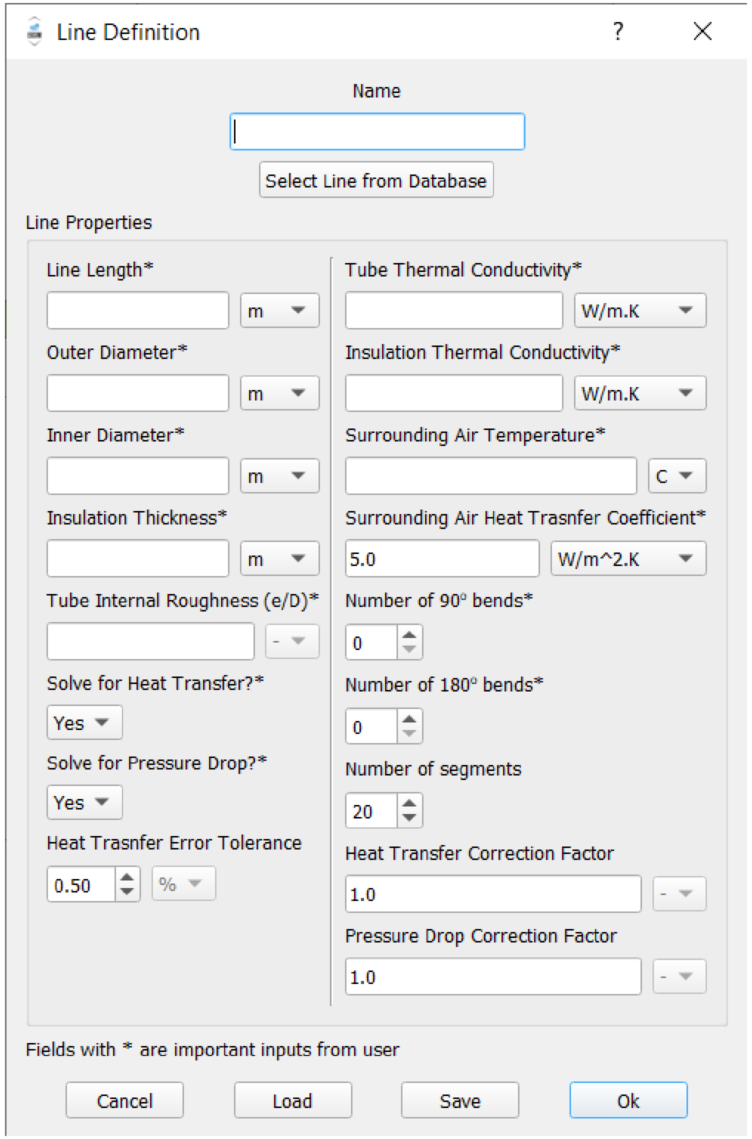
- The user should define the **Insulation Thickness**; if there is no insulation, enter 0
- The user should define **Tube Internal Relative Roughness**, if not known enter 0
- The user should choose to **Solve for Heat Transfer** and/or the **Pressure Drop** by selecting **Yes** in the corresponding dropdown list

If heat transfer is activated, the trial-and-error method will be used for heat transfer calculation, which requires the user to define **Heat Transfer Error Tolerance** criteria



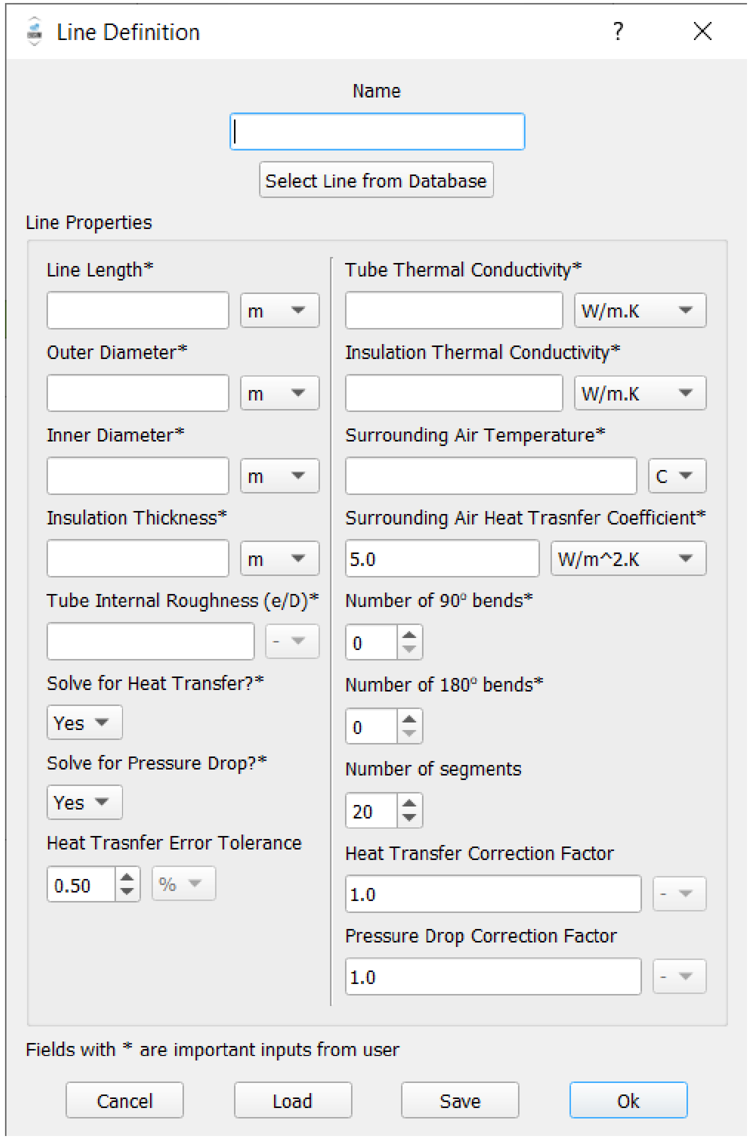
Line Definition, cont.

- The user must specify the **Tube Thermal Conductivity** and the **Insulation Thermal Conductivity**; if there is no insulation use a value of 1.0 for Insulation **Thermal Conductivity**
- The user must define the **Surrounding Air Temperature** which is the temperature around the connecting line and is required for heat transfer calculations
- Finally, the user must assume a value of **Surrounding Air Heat Transfer Coefficient**, which typically range from 5 to 10 W/m^2.K for typical cases



Line Definition, cont.

- If any bends exist in the line, enter the Number of 90° and 180°° bends
- While solving the connecting line, the line will be divided to **Number of Segments** defined by the user; each segment will be solved for heat transfer and pressure drop
- **Heat Transfer and Pressure Drop Correction Factors** can be used to alter the values calculated to empirically validate the model

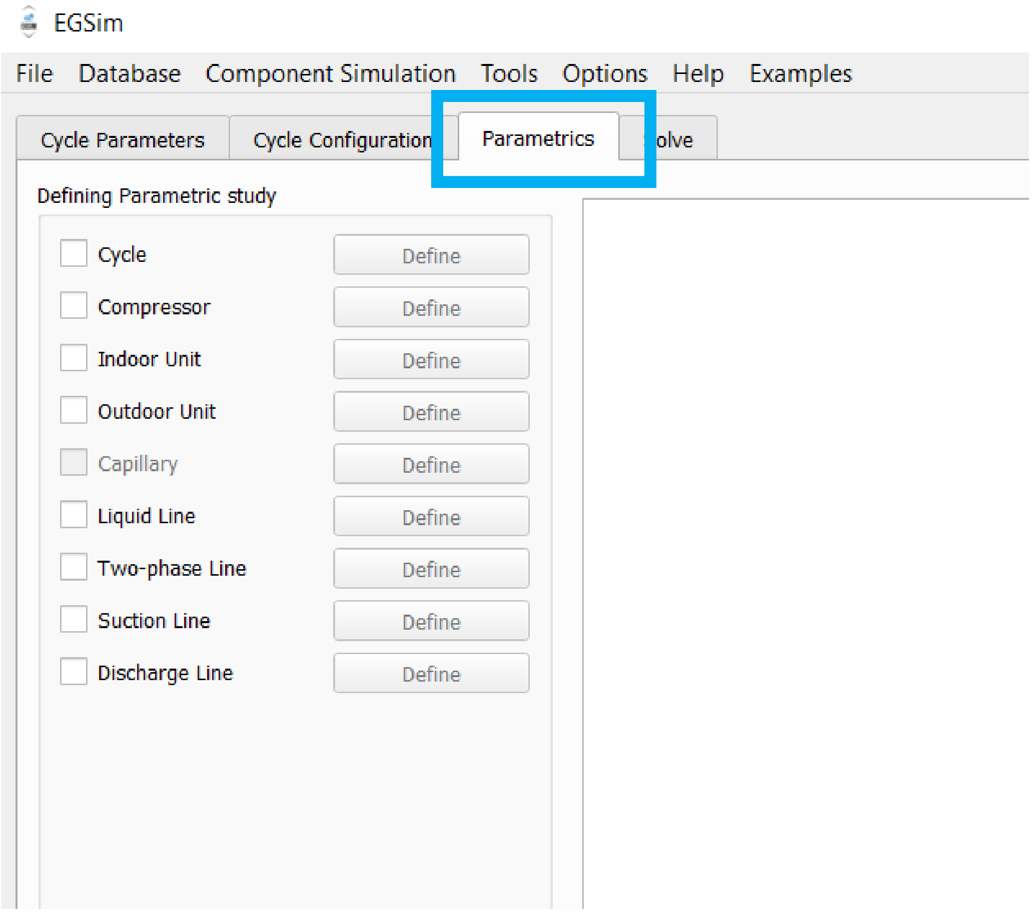


4. Parametric Studies

EGSim User Guide

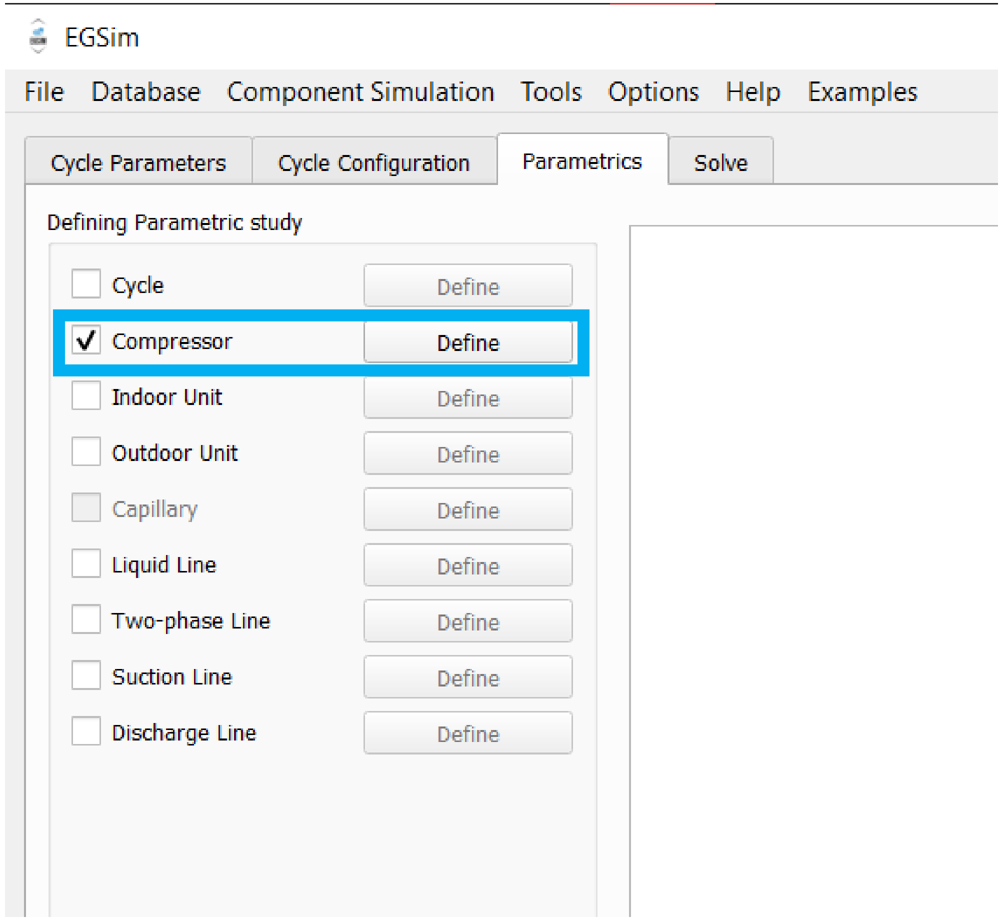
Overview

- The user can define parametric analysis using the **Parametrics** tab to study the impact of different design and operating parameters on system performance
- Parameters can be defined at the following levels:
  - Cycle
  - Indoor unit
  - Outdoor unit
  - Capillary tube (if exists)
  - Connecting lines
- Permutations of values of the parameters are used to generate all the expected configurations or the study



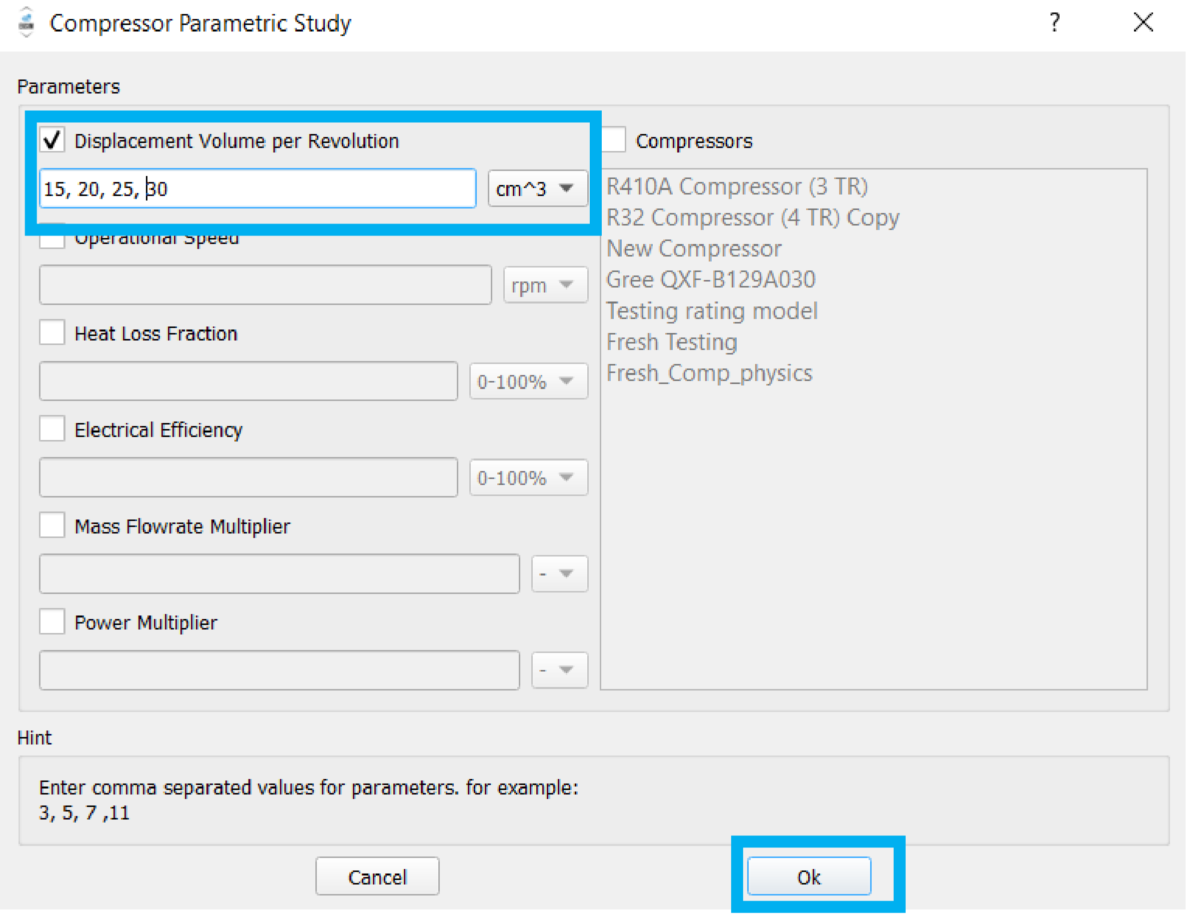
Activating a parametric study

- Once the user places a check on the checkbox next relevant cycle or component, he/she can then click on **Define** button to set the desired values for the parameters to be studied
- Note that in components, you must define the component as shown in section 3 before defining the parametric study



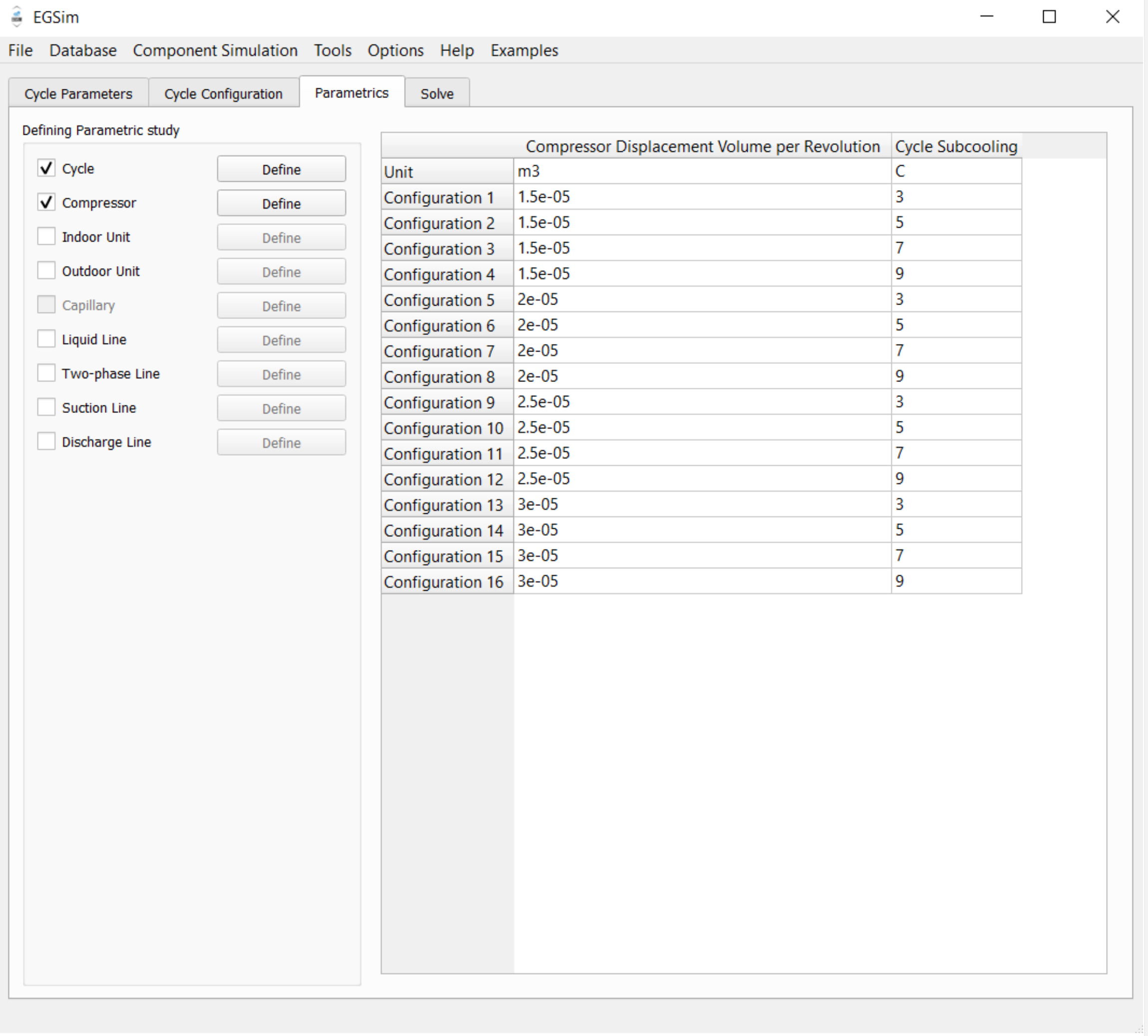
Components parametric study

- A window with available parameters will show up
- The user can select the parameter he/she want to study, by placing a check in the checkbox next to it
- A text box below the activated parameter will be enabled
- The user has to select the desired unit system for the values
- The user enters the values of the parameter to be studied with a "," separating the values
- Example input: 15,20,25,30
- Finally, press **OK**



Several parameters

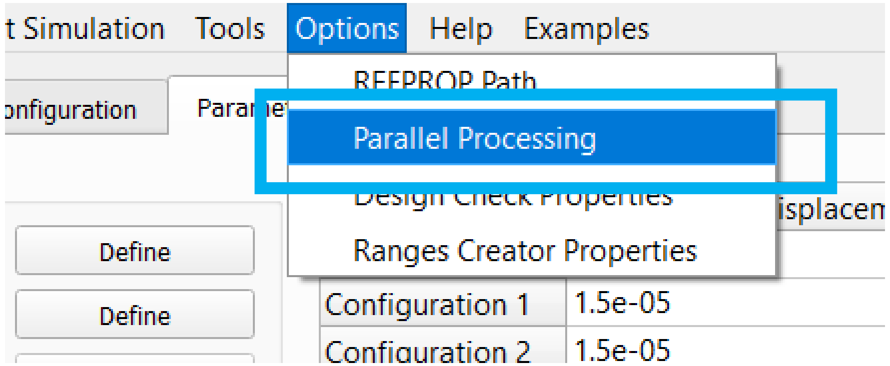
- Once the user press ok, a table will appear showing the studied parameters in columns, and the different permutations as rows.
- The total number of configurations will be the multiple of the number of values for each parameter
- here we have 2 parameters each having 4 values, so 4*4 = 16 configurations

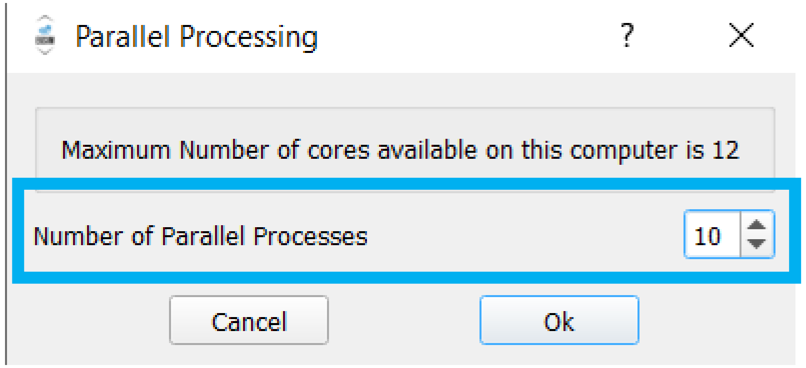


	Compressor Displacement Volume per Revolution	Cycle Subcooling
Unit	m3	C
Configuration 1	1.5e-05	3
Configuration 2	1.5e-05	5
Configuration 3	1.5e-05	7
Configuration 4	1.5e-05	9
Configuration 5	2e-05	3
Configuration 6	2e-05	5
Configuration 7	2e-05	7
Configuration 8	2e-05	9
Configuration 9	2.5e-05	3
Configuration 10	2.5e-05	5
Configuration 11	2.5e-05	7
Configuration 12	2.5e-05	9
Configuration 13	3e-05	3
Configuration 14	3e-05	5
Configuration 15	3e-05	7
Configuration 16	3e-05	9

Multi-processing

- EGSim can run several configurations simultaneously using available processor cores
- From **Options** => **Parallel Processing**, the user can define the number of configurations to be solved simultaneously
- The window will also show the available number of cores on the computer
- The **Number of Parallel Processes** should be less than or equal the number of available cores
- It is advised not to use all available cores to avoid freezing



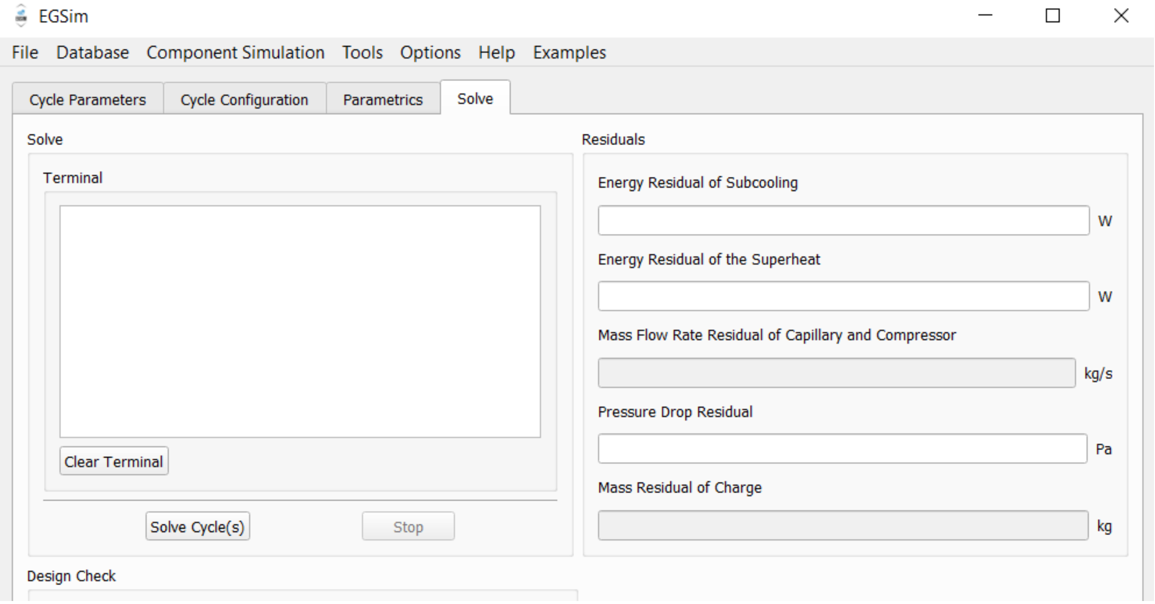


5. Solving Cycle

EGSim User Guide

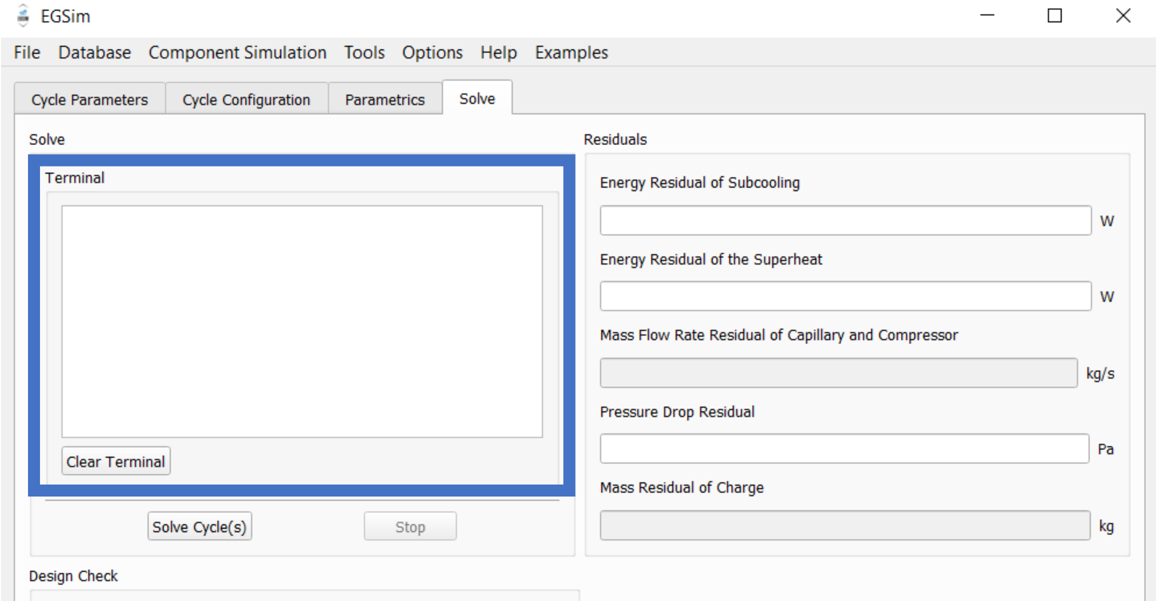
Overview

- Once the cycle is fully defined, the user can solve the cycle
- Solving time will depend on the inputs
  - Segment-by-segment heat exchangers will take longer time to solve than phase-by-phase heat exchangers
  - Initial guess for condenser and evaporator temperature can also affect the cycle solution time



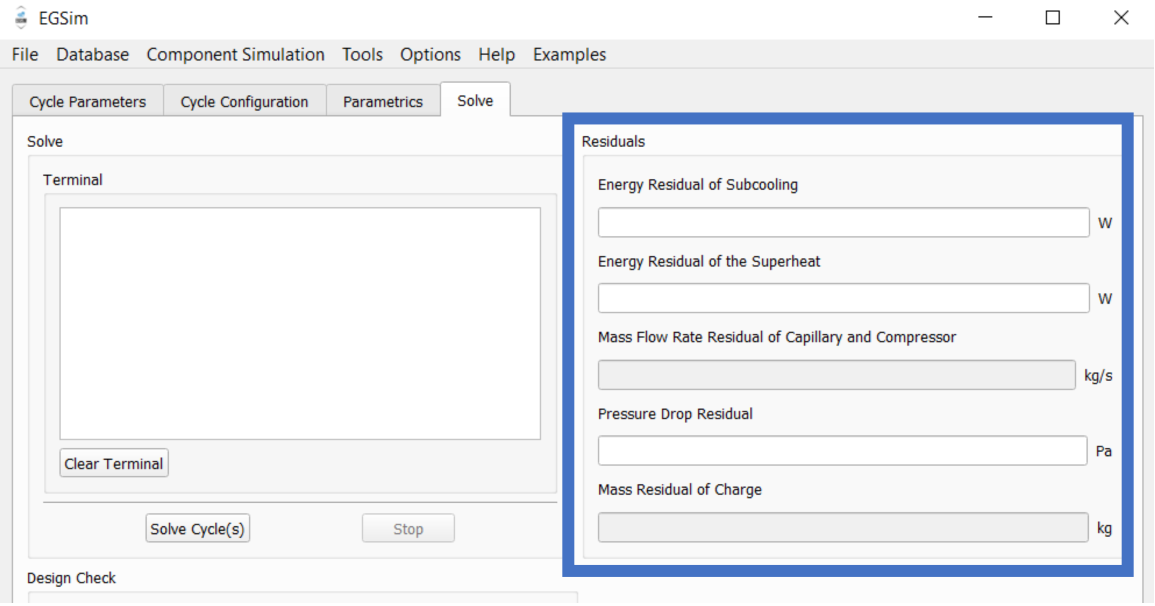
Terminal

- In the terminal, you will find relevant information about solving the cycle.
- You can clear the terminal messages by clicking **Clear Terminal**



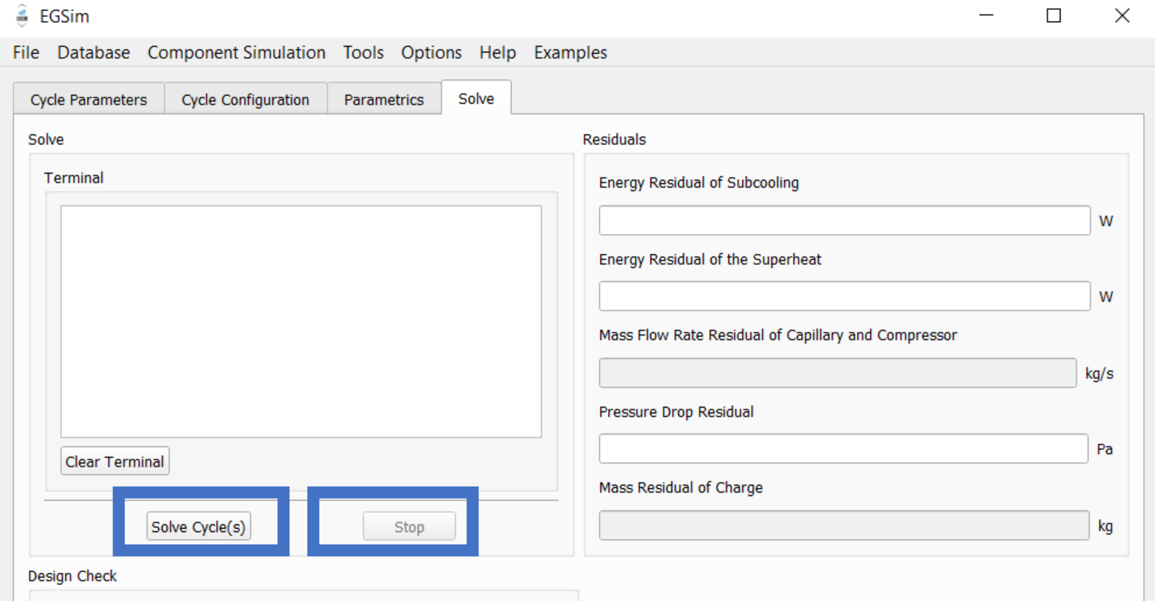
Residuals

- If the user is not performing a parametric study, he/she will find some of the residuals activated, depending on the **Solving for?** options selected in **Cycle Parameters**
- **Residuals** the cycle solution progress
- The solver is converged when **Residuals** are less than the chosen criteria in **Cycle Parameters**
- In **Parametrics** study, each configuration will have its own residuals, so residuals will not be shown – however, the solution progress will appear in the terminal



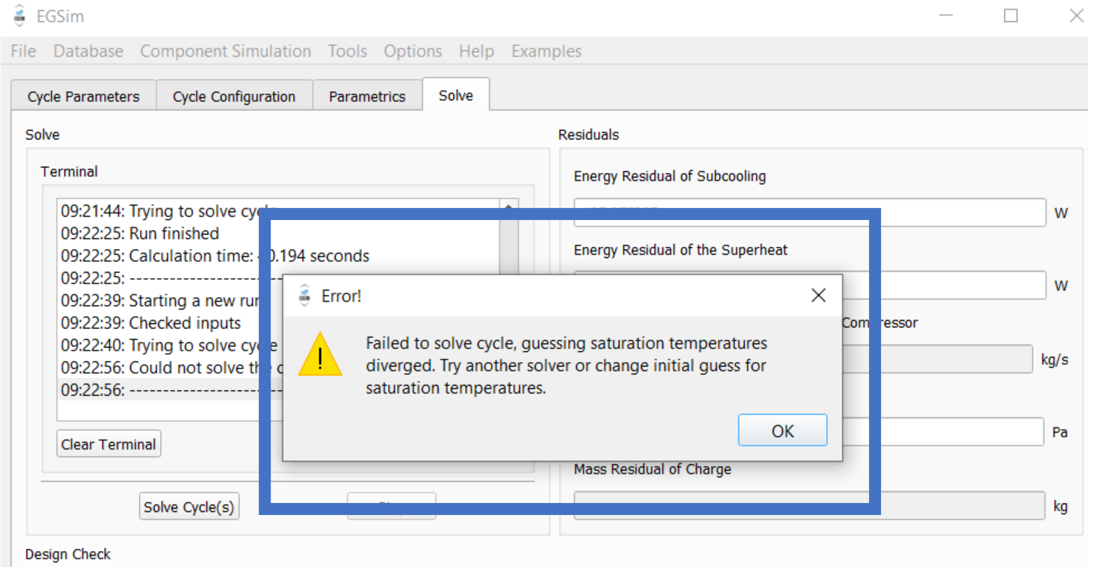
Solve Cycle

- To initiate solving the cycle, the user has to click on **Solve Cycle(s)** button
- The user can stop the current solution to edit the cycle, or if the solution is taking extended time and by clicking on the **Stop** button



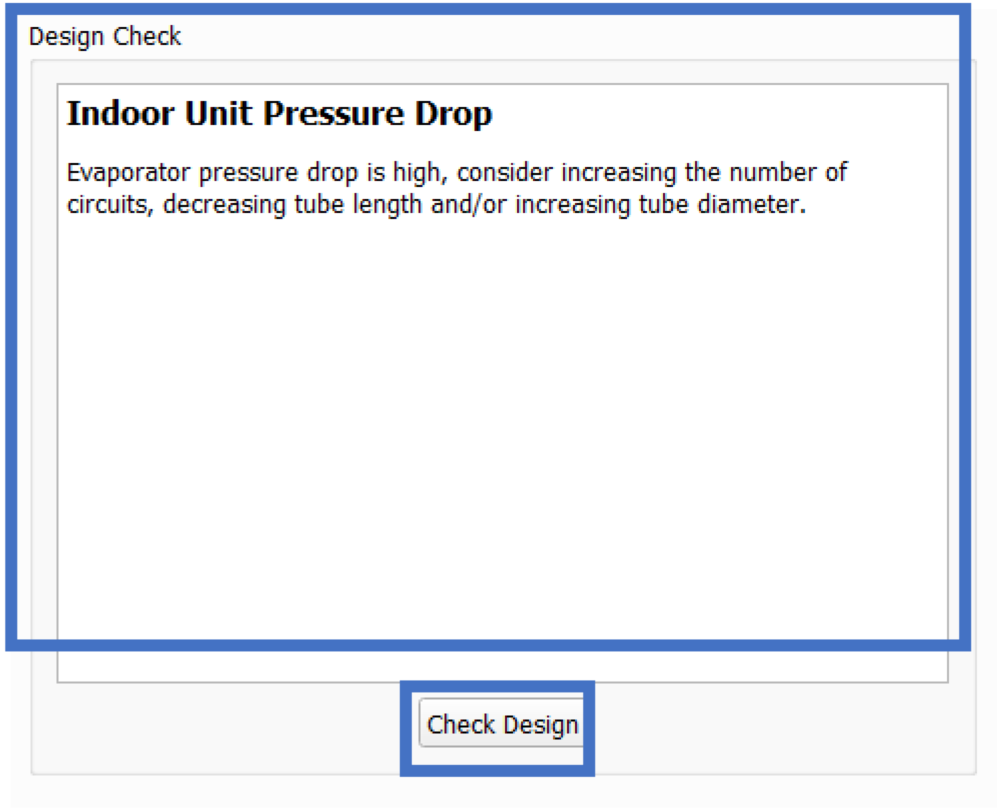
Solve Cycle, cont.

- If the solver fails to solve the cycle, an error window will appear, indicating a failed cycle solution, with the probable cause for the failure



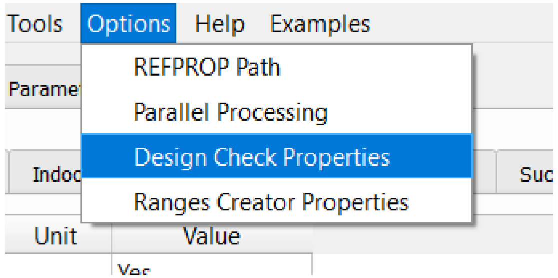
Design Check

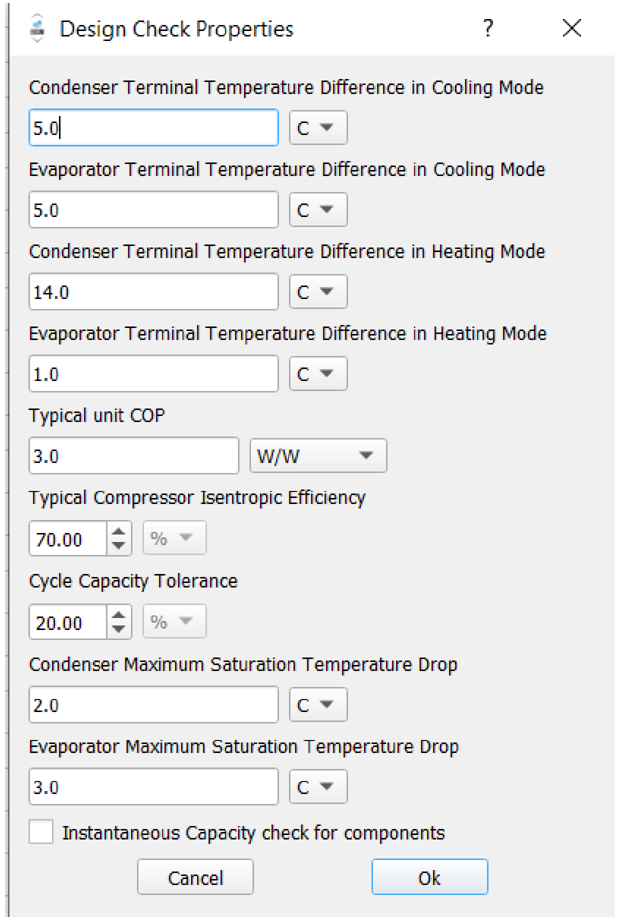
- The user may also check his/her design for recommendations by in the **Design Check** terminal by clicking on **Check Design** button
- The Design Check feature will perform
  - A check on the heat exchangers capacity against the target capacity and pressure drop on refrigerant side
  - A check on compressor for capacity against target capacity



Design Check, cont.

- Design check settings can be changed through **Options=>Design Check Properties**
- If the user wants to have instantaneous Design Check as he/she defines the components, the user has to place a check on the checkbox **Instantaneous Capacity check for components**

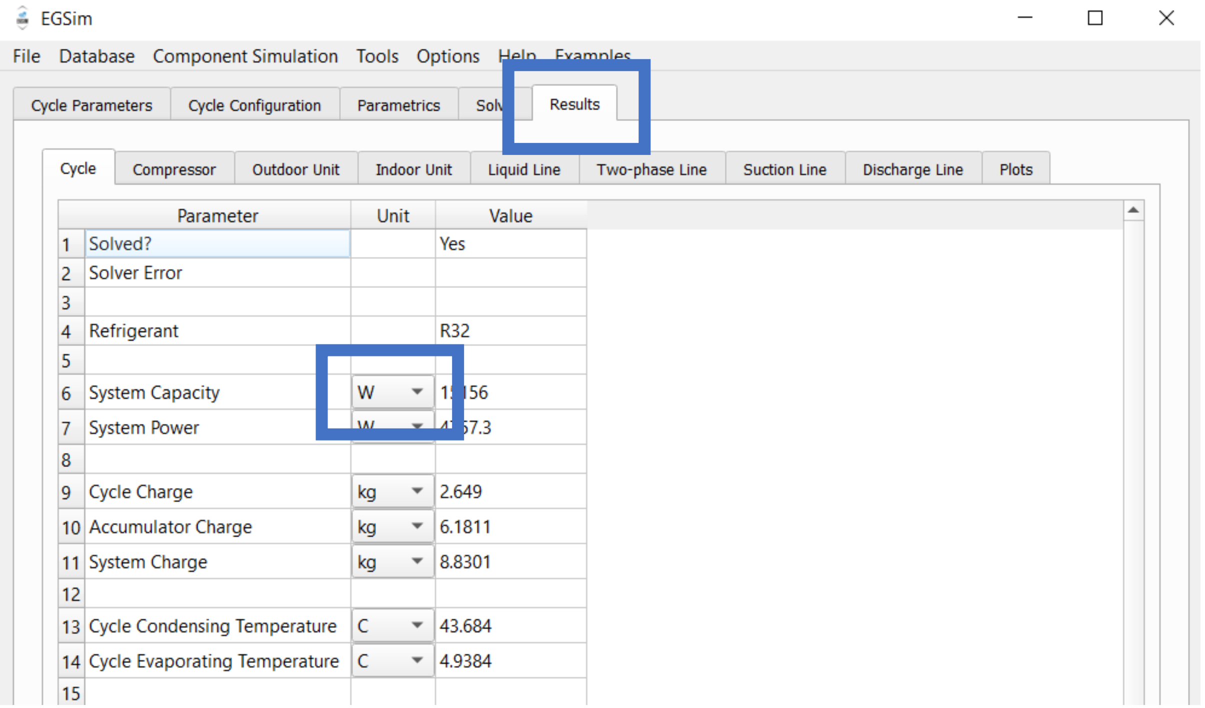




6. Results

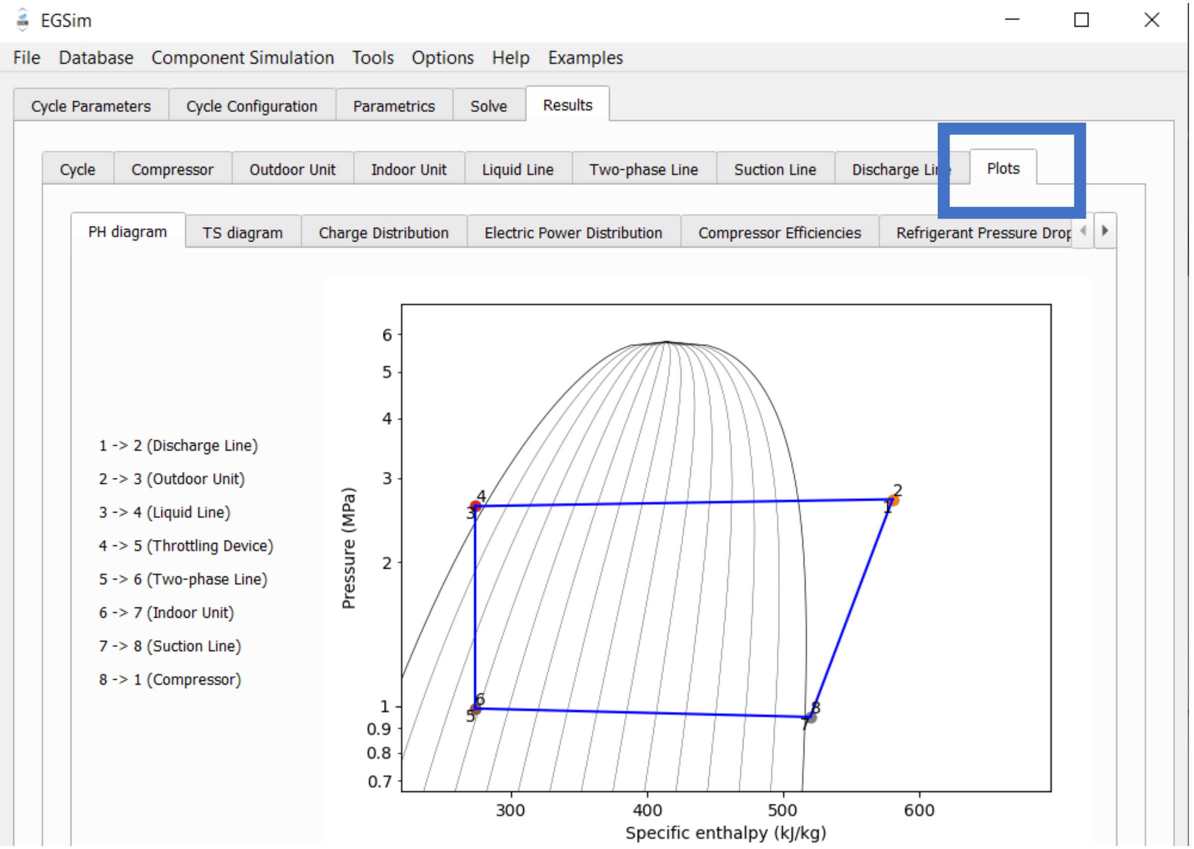
Overview

- Once the cycle is solved successfully, the **Results** tab will appear showing the detailed solution results for the
  - Cycle
  - Components, each on a separate tab
  - Relevant plots
- The user can change the results unit as shown in the dropdown list menu next to any desired output



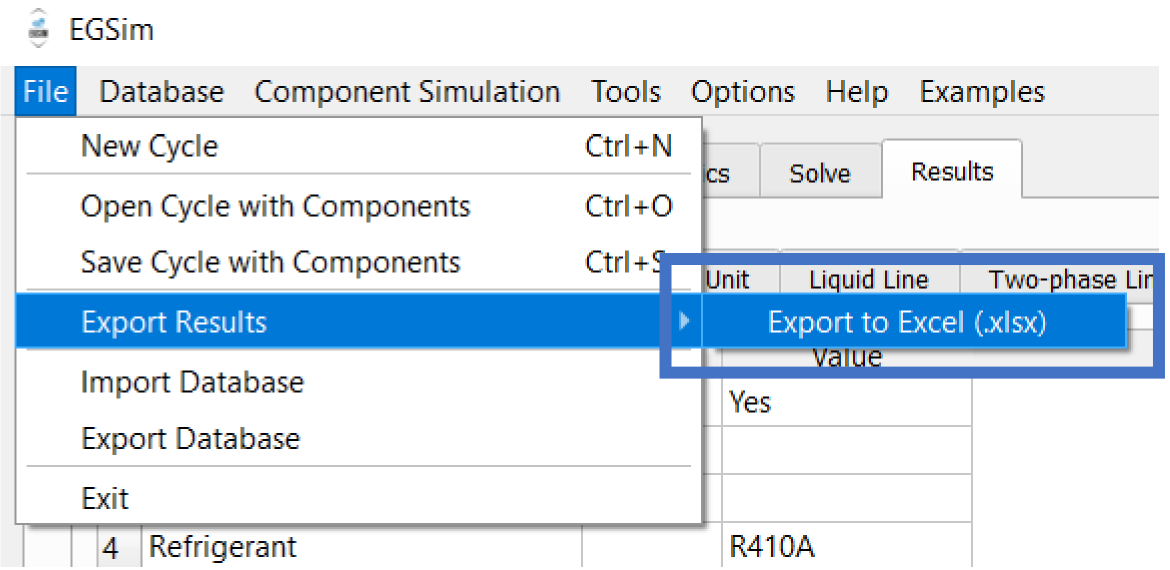
Plots

- The **Plots** tab includes
  - P-H and T-S Diagrams
  - Charge Distribution
  - Electric Power Distribution
  - Compressor Efficiencies
  - Refrigerant Pressure Drop Distribution
  - Entropy Generation
  - Outdoor/Indoor Unit
    - Capacity Distribution
    - Refrigerant Pressure Drop Distribution
    - Heat Exchanger Area Distribution
    - Refrigerant Heat Transfer Coefficients (Ref HTC)



Export Results

- The user can export the results externally to an excel file through **File=>Export Results=>Export to Excel (.xlsx)** to allow for further analysis as needed

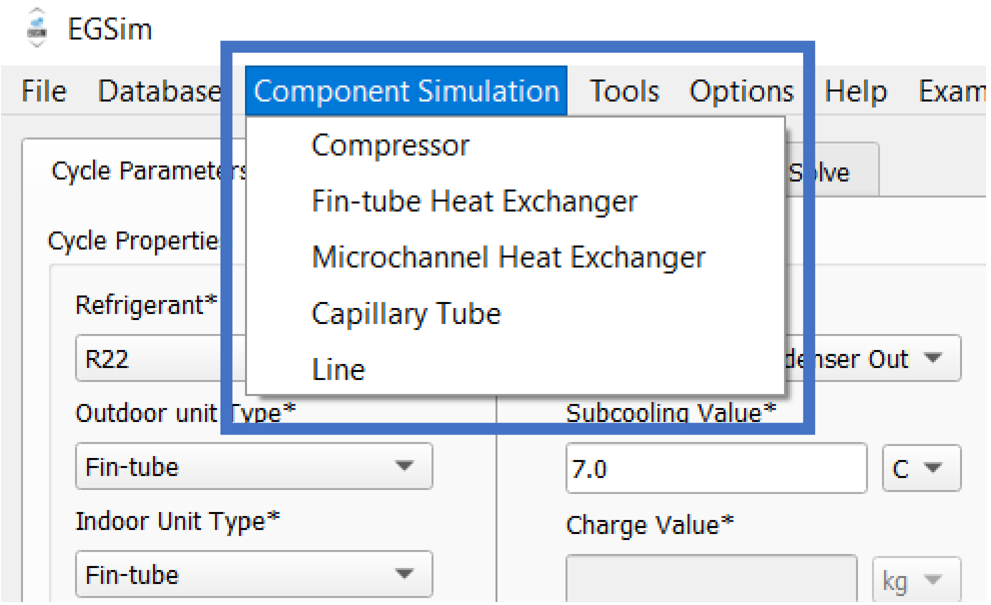


7. Components Simulation

EGSim User Guide

Overview

- You can simulate a single component by its own through the **Components Simulation** menu
- The performance of relevant components can be simulated without the need to define the full cycle, however, the user will have to provide relevant inputs



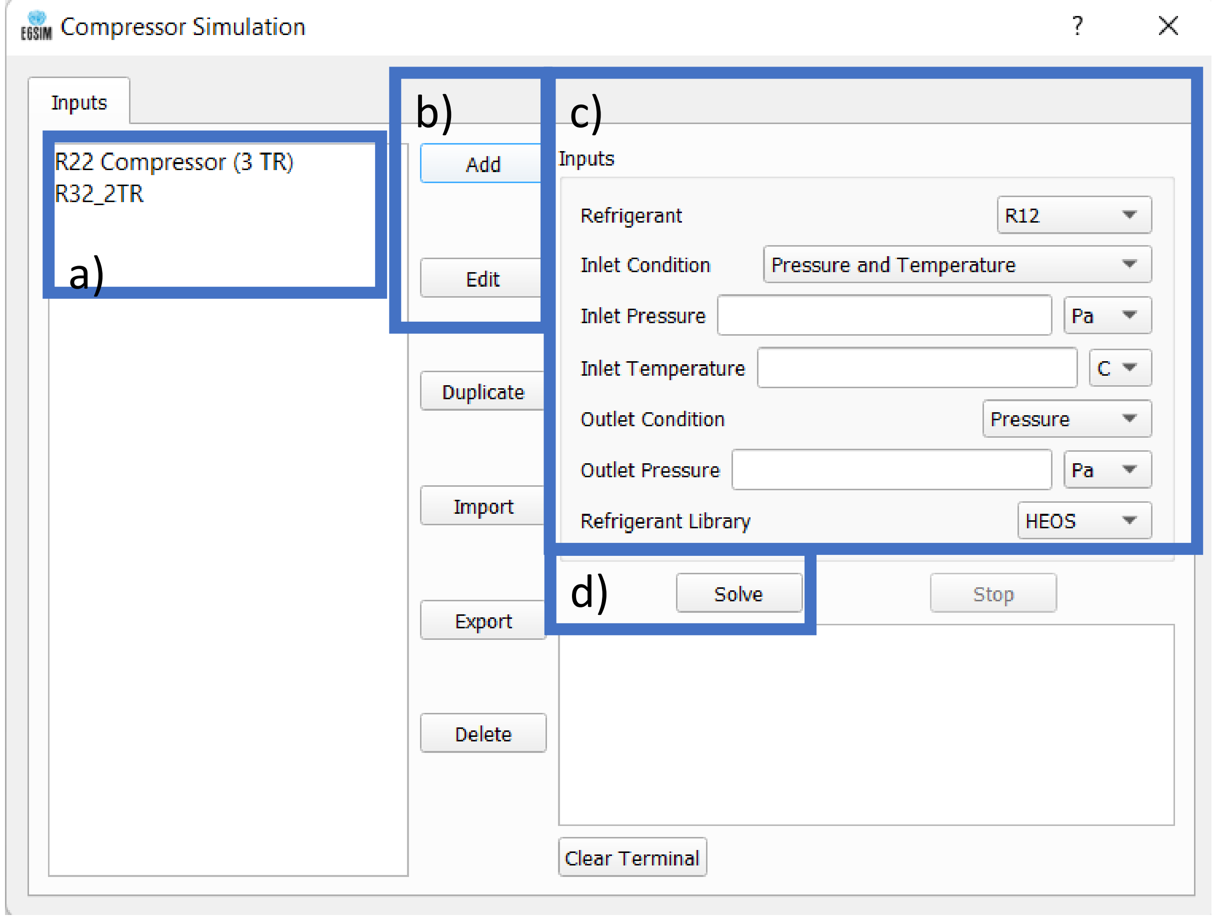
Compressor

a) To simulate a **Compressor**, first choose a **Compressor** from the **Database**

b) The user can **Edit** or **Add** compressors as needed – similar to the process defined in section 3.1

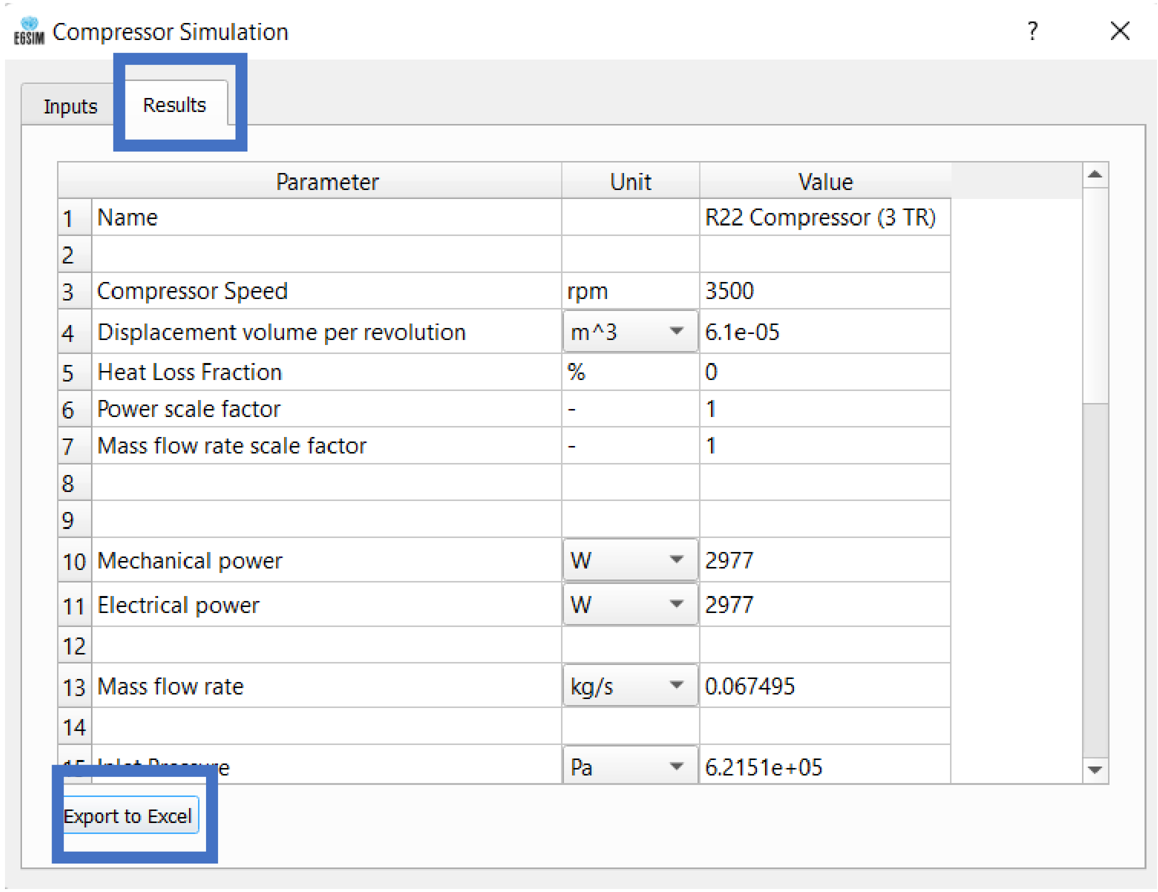
c) In the **Inputs** section, the user will need to choose the **Refrigerant**, the **Inlet Condition**, the **Outlet Condition**, and the **Refrigerant Library**

d) Finally click **Solve** to simulate the component



Compressor, cont.

- Once the **Compressor** and relevant inputs are defined, the user can click **Solve** to get the compressor performance
- A new **Results** tab will appear showing the compressor performance
- The user can export the results to excel by clicking on Export to Excel Button



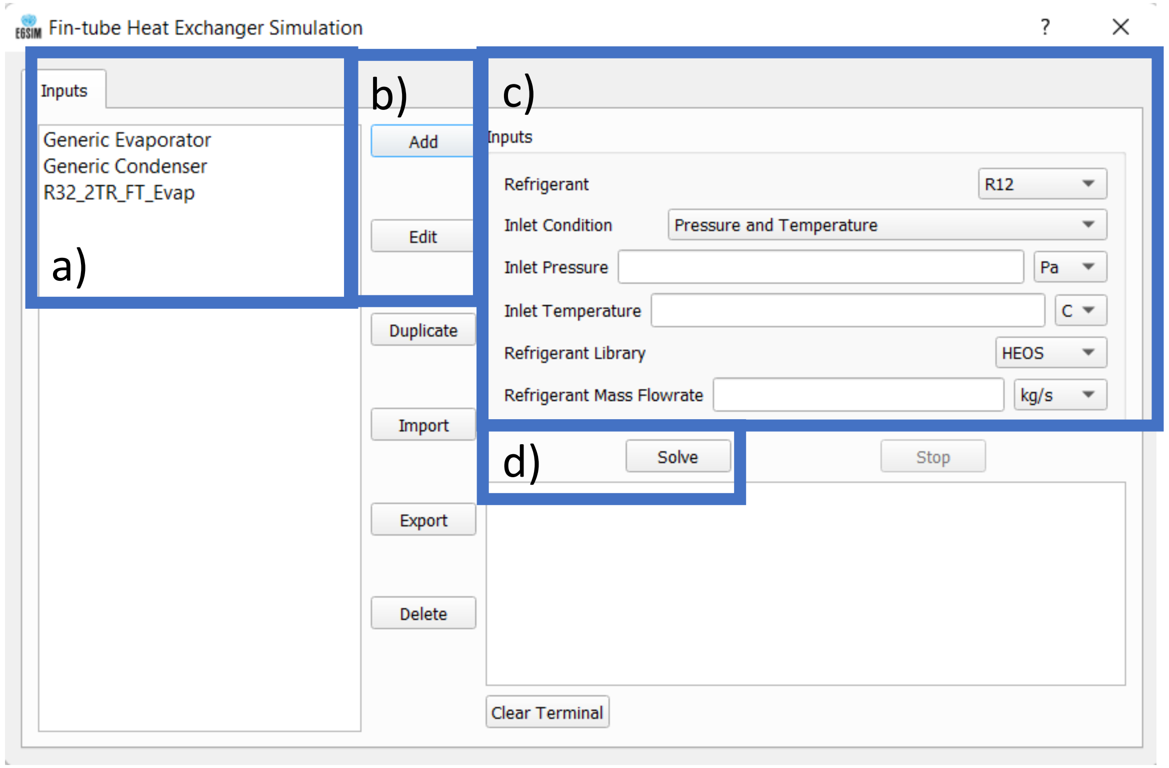
Fin-tube Heat Exchanger

a) To simulate a **Fin-tube Heat Exchanger**, first choose a **Fin-tube Heat Exchanger** from the **Database**

b) The user can **Edit** or **Add Fin-tube Heat Exchanger** as needed – similar to the process defined in section 3.2

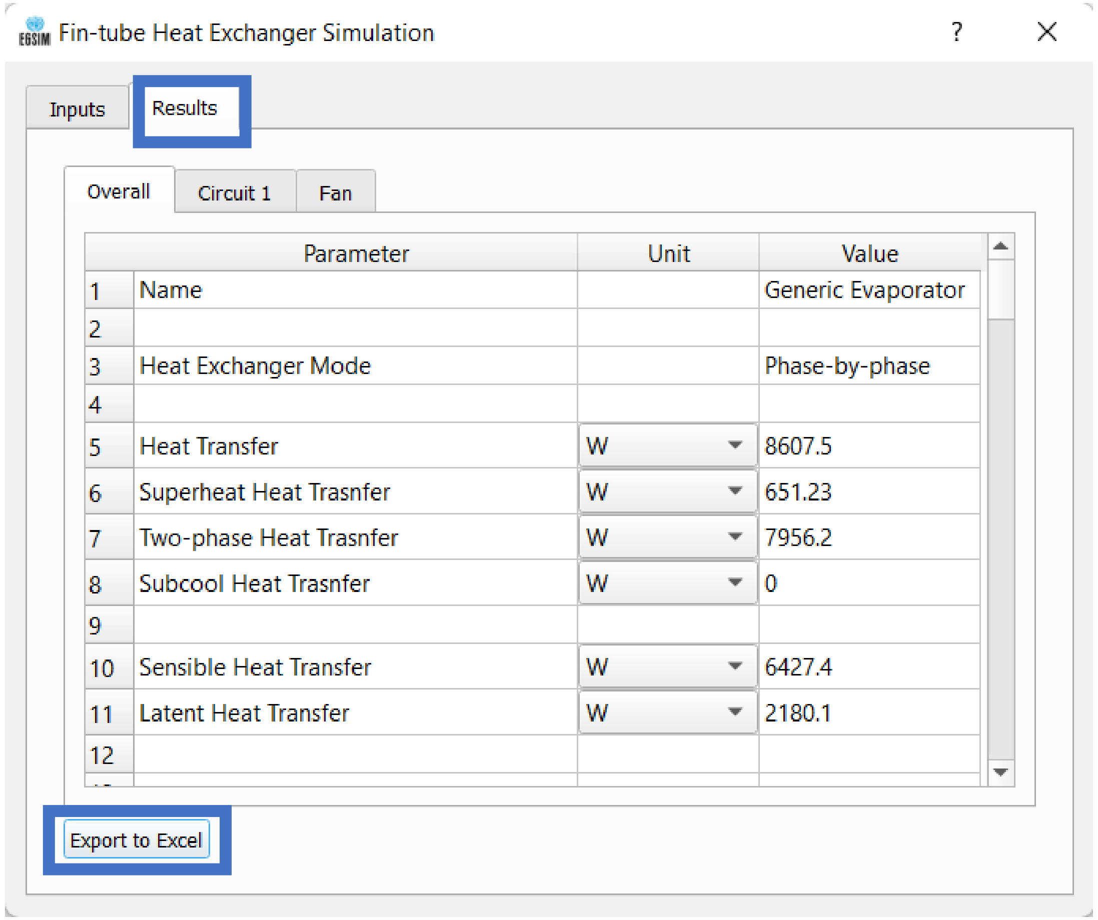
c) In the **Inputs** section, the user will need to choose the **Refrigerant**, the **Inlet Condition**, the **Refrigerant Library,** and the **Refrigerant Mass Flowrate**

d) Finally click **Solve** to simulate the component



Fin-tube Heat Exchanger, cont.

- Once the **Fin-tube Heat Exchanger** and relevant inputs are defined, the user can click **Solve** to get the heat exchanger performance
- A new **Results** tab will appear showing the heat exchanger performance
- The user can export the results to excel by clicking on **Export to Excel** Button



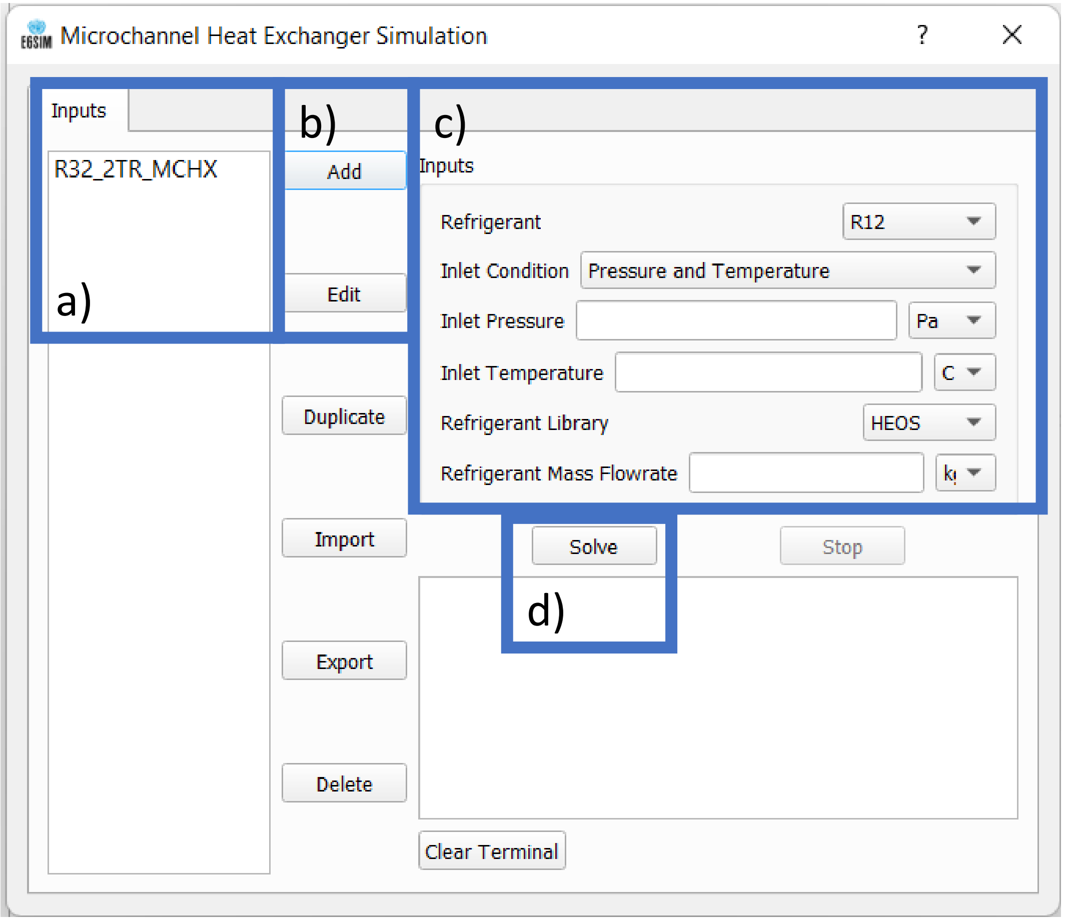
Microchannel Heat Exchanger

a) To simulate a **Microchannel Heat Exchanger**, first choose a **Microchannel Heat Exchanger** from the **Database**

b) The user can **Edit** or **Add Microchannel Heat Exchanger** as needed – similar to the process defined in section 3.3

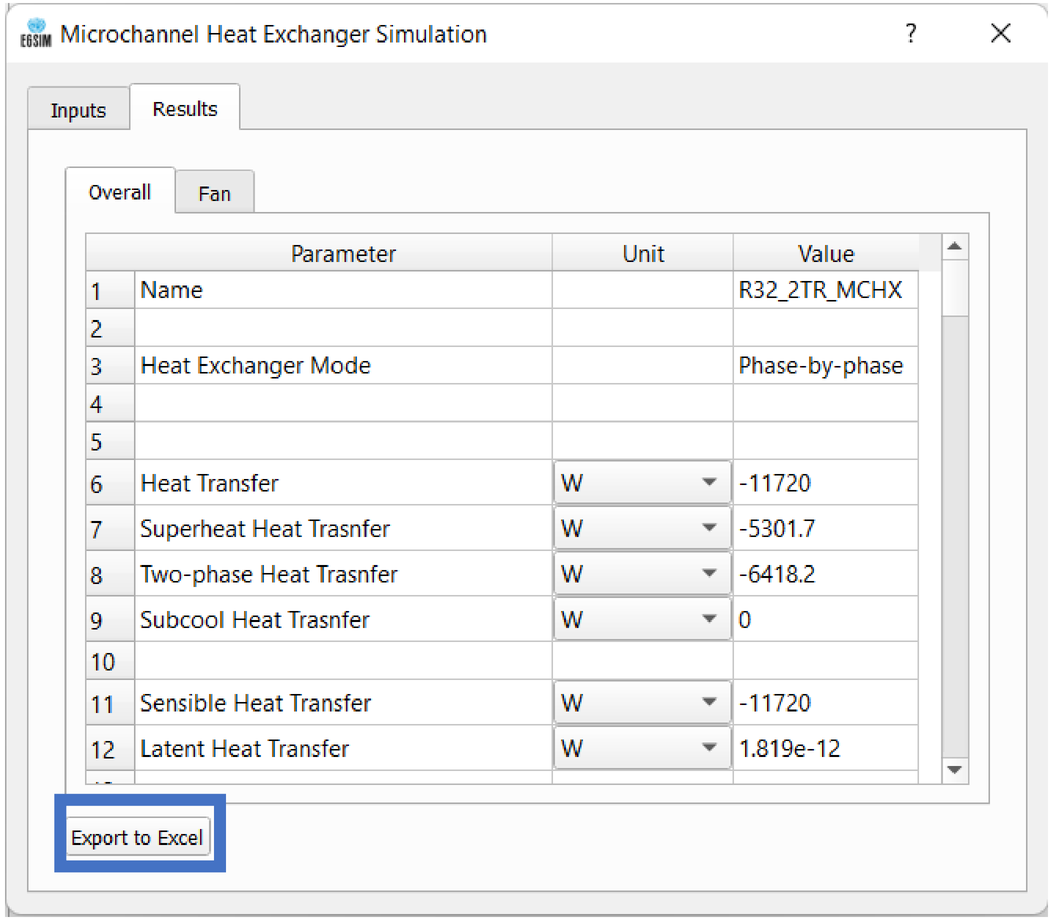
c) In the **Inputs** section, the user will need to choose the **Refrigerant**, the **Inlet Condition**, the **R efrigerant Library,** and the **Refrigerant Mass Flowrate**

d) Finally click **Solve** to simulate the component



Microchannel Heat Exchanger, cont.

- Once the **Microchannel Heat Exchanger** and relevant inputs are defined, the user can click **Solve** to
  get the heat exchanger performance
- A new **Results** tab will appear showing the heat exchanger performance
- The user can export the results to excel by clicking on **Export to Excel** Button



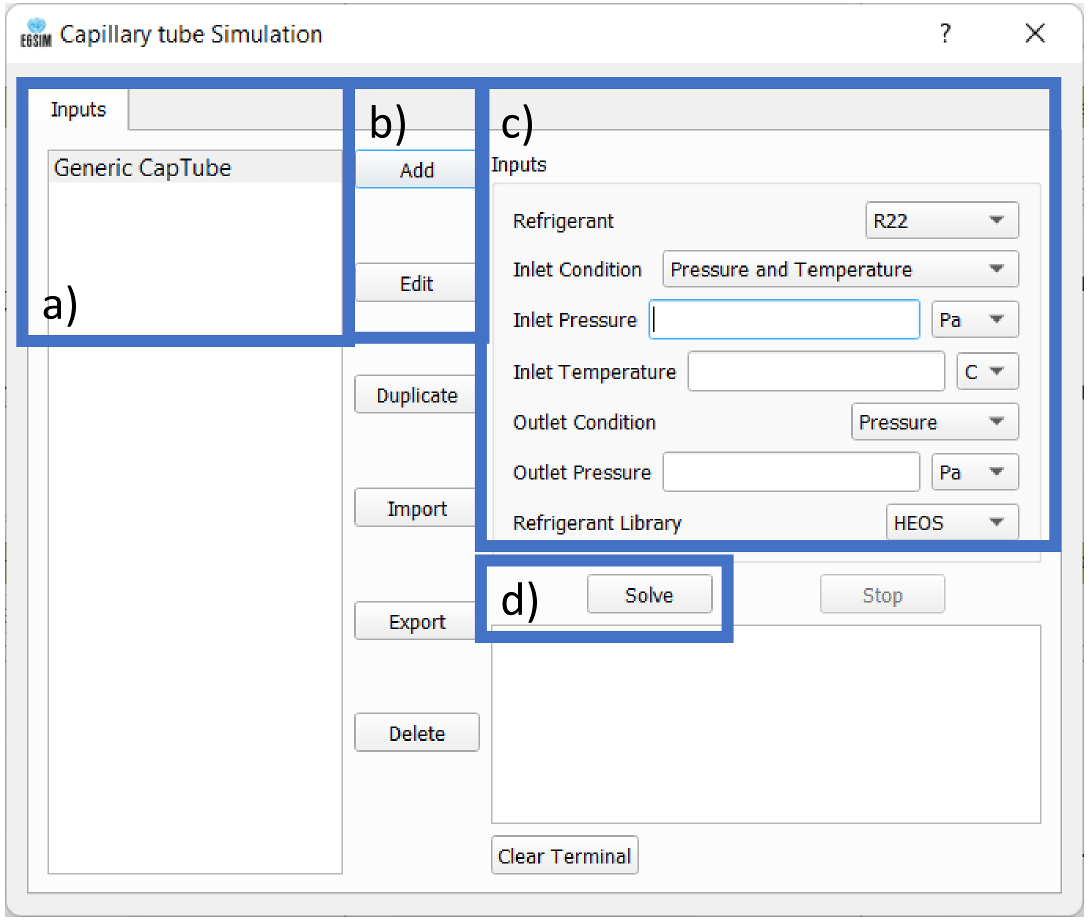
Capillary Tube

a) To simulate a **Capillary Tube**, first choose a **Capillary Tube** from the **Database**

b) The user can **Edit** or **Add Capillary Tube** as needed

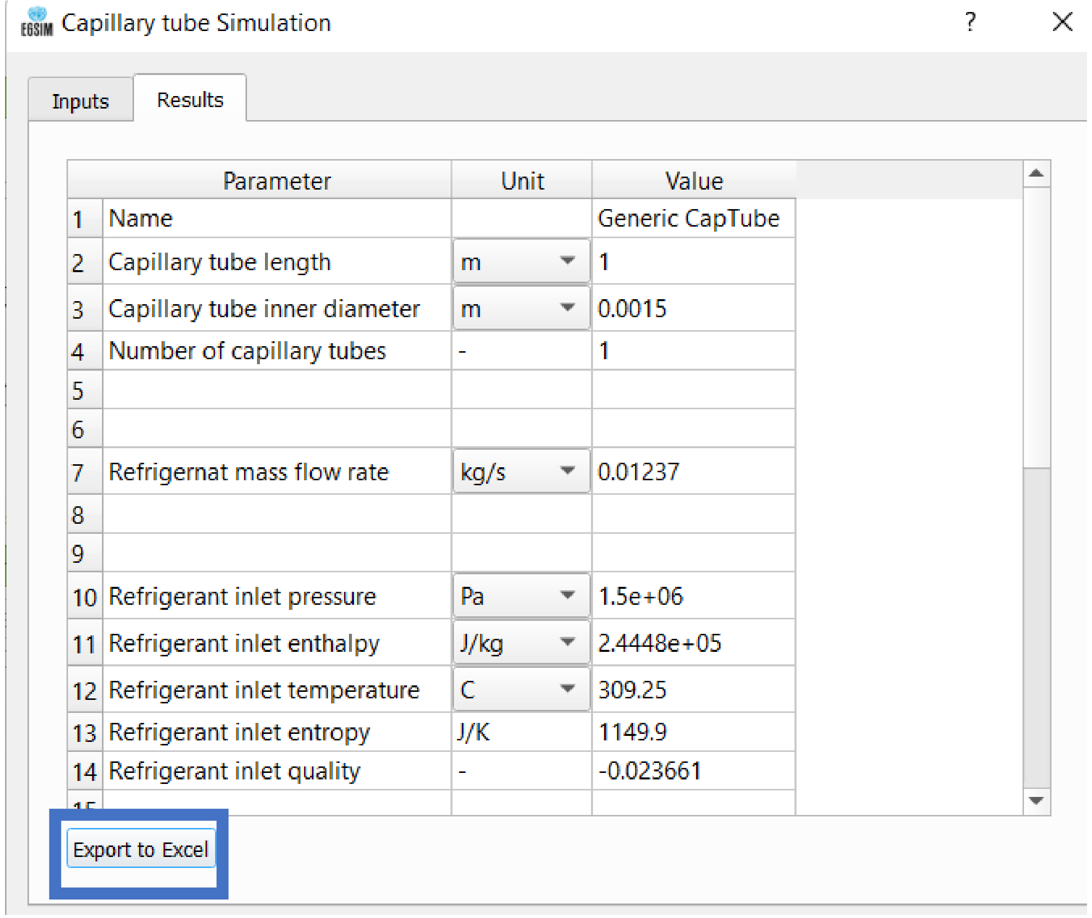
c) In the **Inputs** section, the user will need to choose the **Refrigerant**, the **Inlet Conditi on**, the **Outlet Condition**, and the **Refrigerant Library**

d) Finally click **Solve** to simulate the component



Capillary Tube, cont.

- Once the **Capillary Tube** and relevant inputs are defined, the user can click **Solve** to find the length of the **Capillary Tube** Required
- A new **Results** tab will appear showing the heat exchanger performance
- The user can export the results to excel by clicking on **Export to Excel** Button



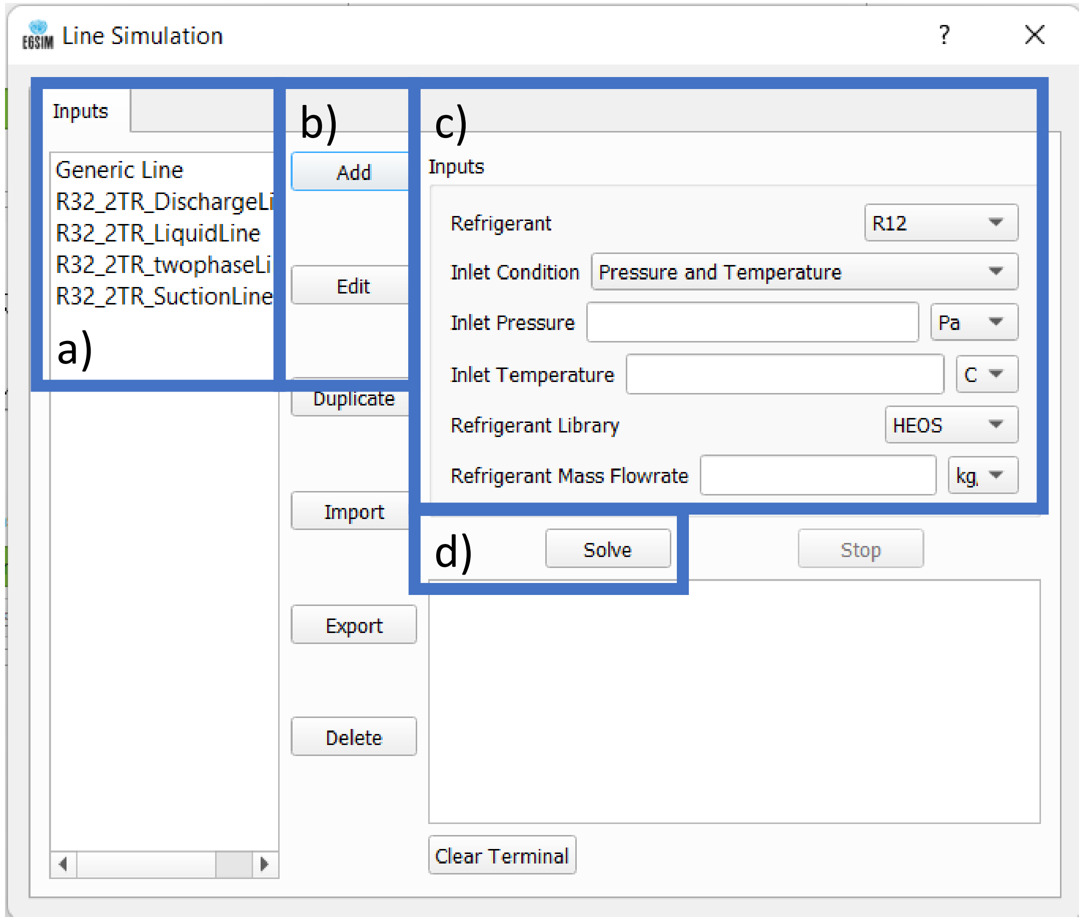
Connecting Line

a) To simulate a **Connecting Line**, first choose a **Connecting Line** from the **Database**

b) The user can **Edit** or **Add Connecting Line** as needed – similar to the process described in section 3.4

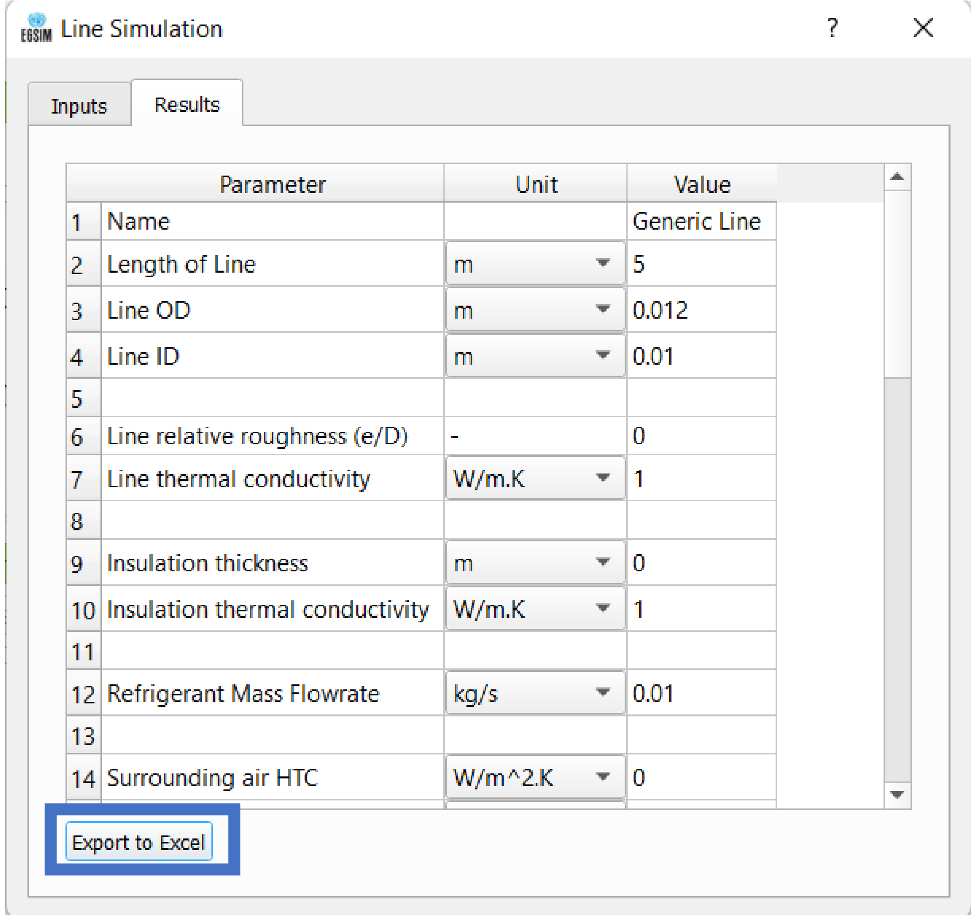
c) In the **Inputs** section, the user will need to choose the **Refrigerant**, the **Inlet Condition**, the **Refrigerant Library**, and the **Refrigerant Mass Flowrate**

d) Finally click **Solve** to simulate the component



Connecting Line, cont.

- Once the **Connecting Line** and relevant inputs are defined, the user can click **Solve** to get line performance
- A new **Results** tab will appear showing the performance details
- The user can export the results to excel by clicking on **Export to Excel** Button

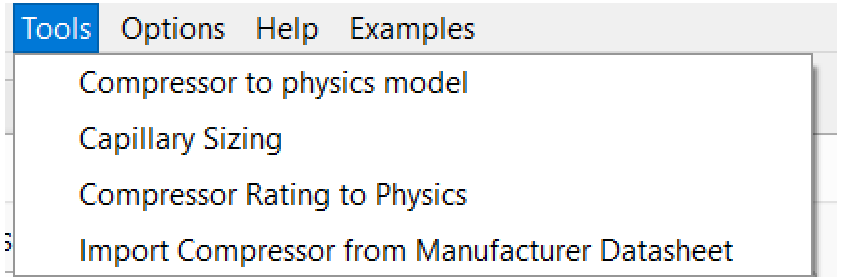


8. Tools

EGSim User Guide

Overview

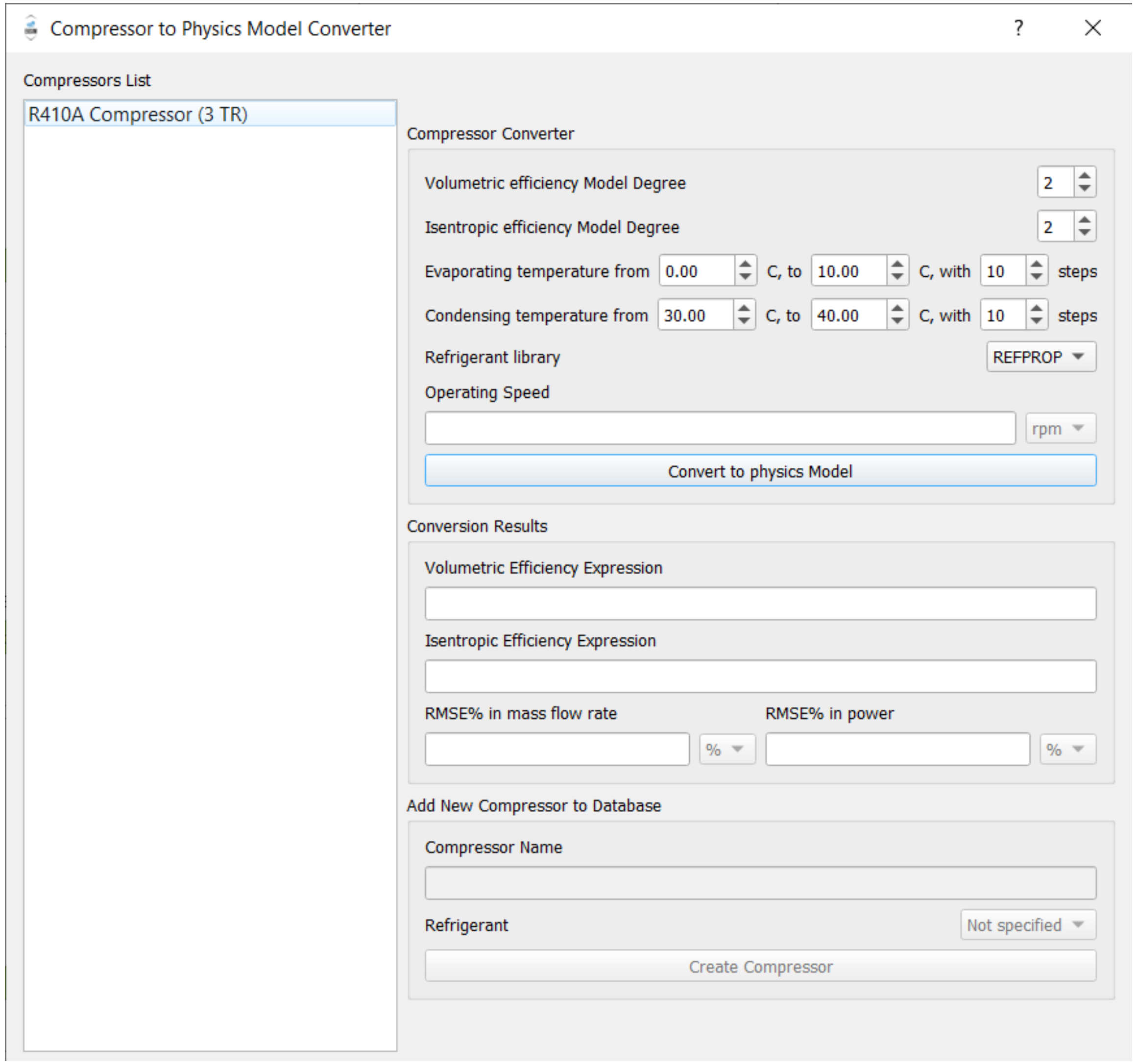
- The **Tools** Menu contain additional functionalities that can be used along with EGSim
- Current Tools include:
  - Compressor to physics model
  - Capillary Sizing
  - Compressor Rating to Physics
  - Import Compressor from Manufacturer Datasheet



Compressor to Physics Model

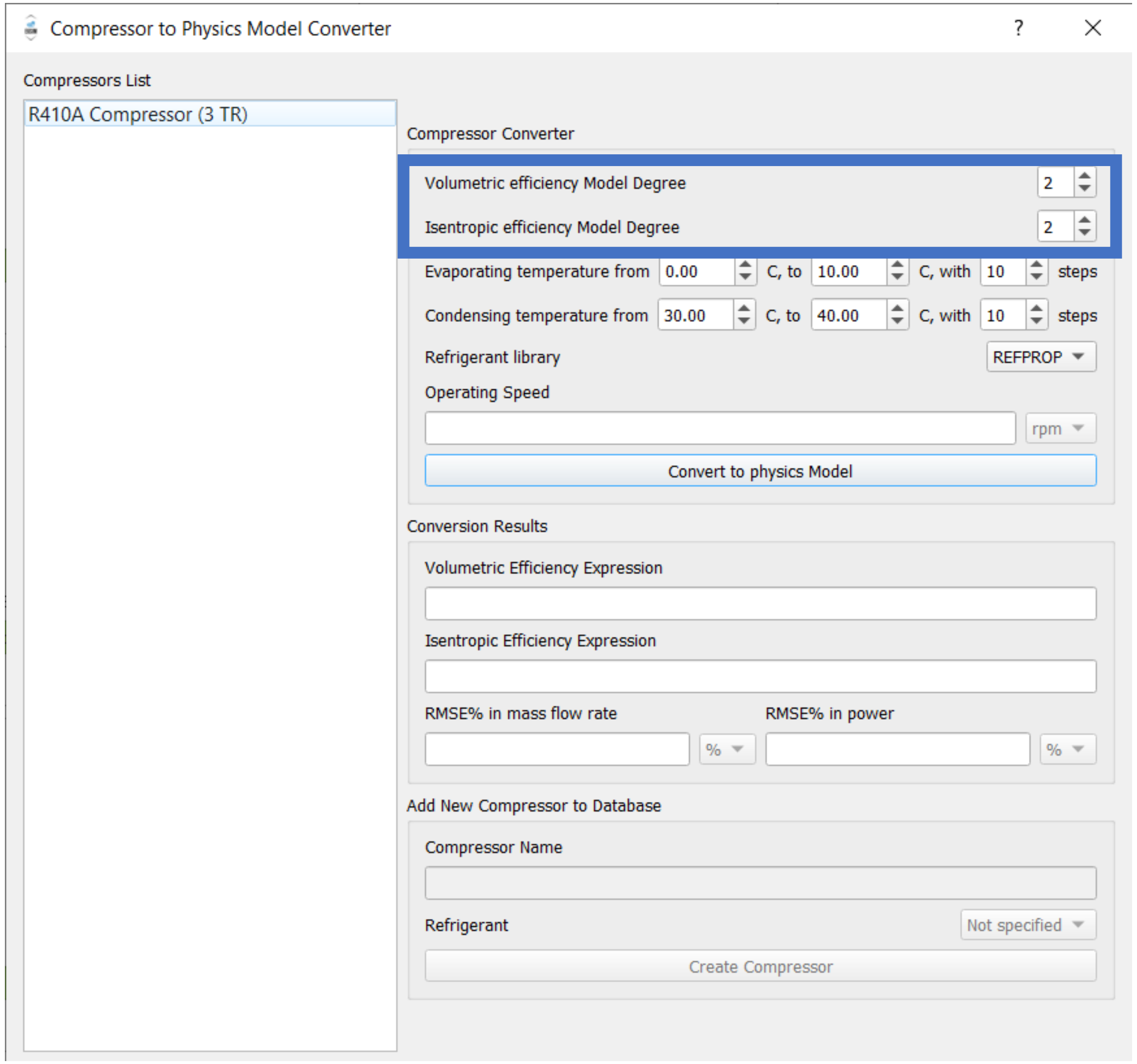
Compressor to Physics Model Converter

- This tool is used to convert a compressor defined as a 10-Coefficients or a Compressor Map to a Physics-based model
- This is helpful in case the user wants to simulate a compressor performance with refrigerants different than the designated refrigerant used to develop the compressor performance datasheet



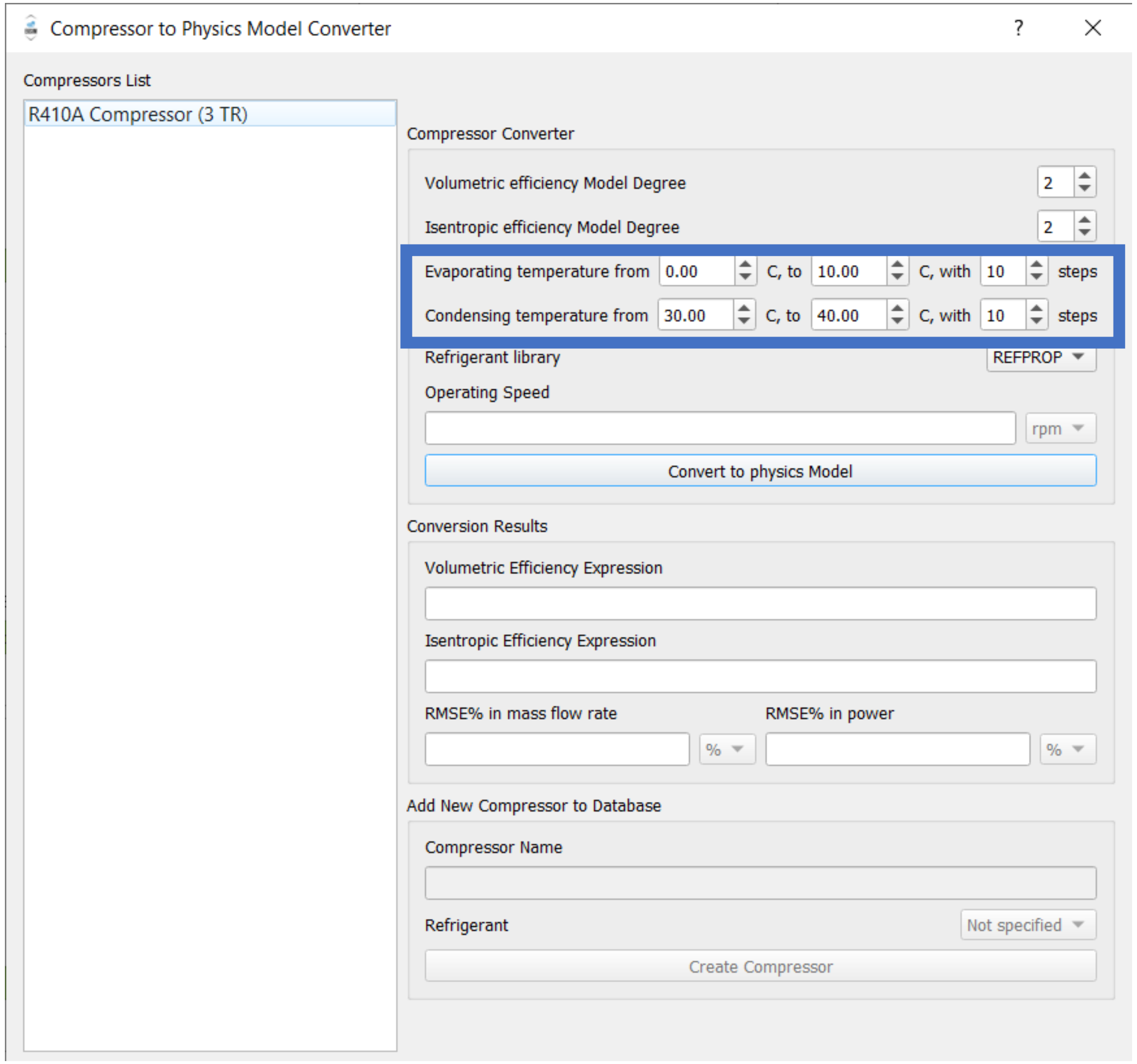
Compressor to Physics Model Converter, cont.

- From the left list, choose the compressor that you wish to convert
- The Physics-based model will be based on polynomial functions of the pressure ratio for the isentropic and volumetric efficiencies
- The user can choose the degree of the polynomial function for isentropic efficiencies and volumetric efficiency, a third order polynomial is typically sufficient to provide the desired accuracy



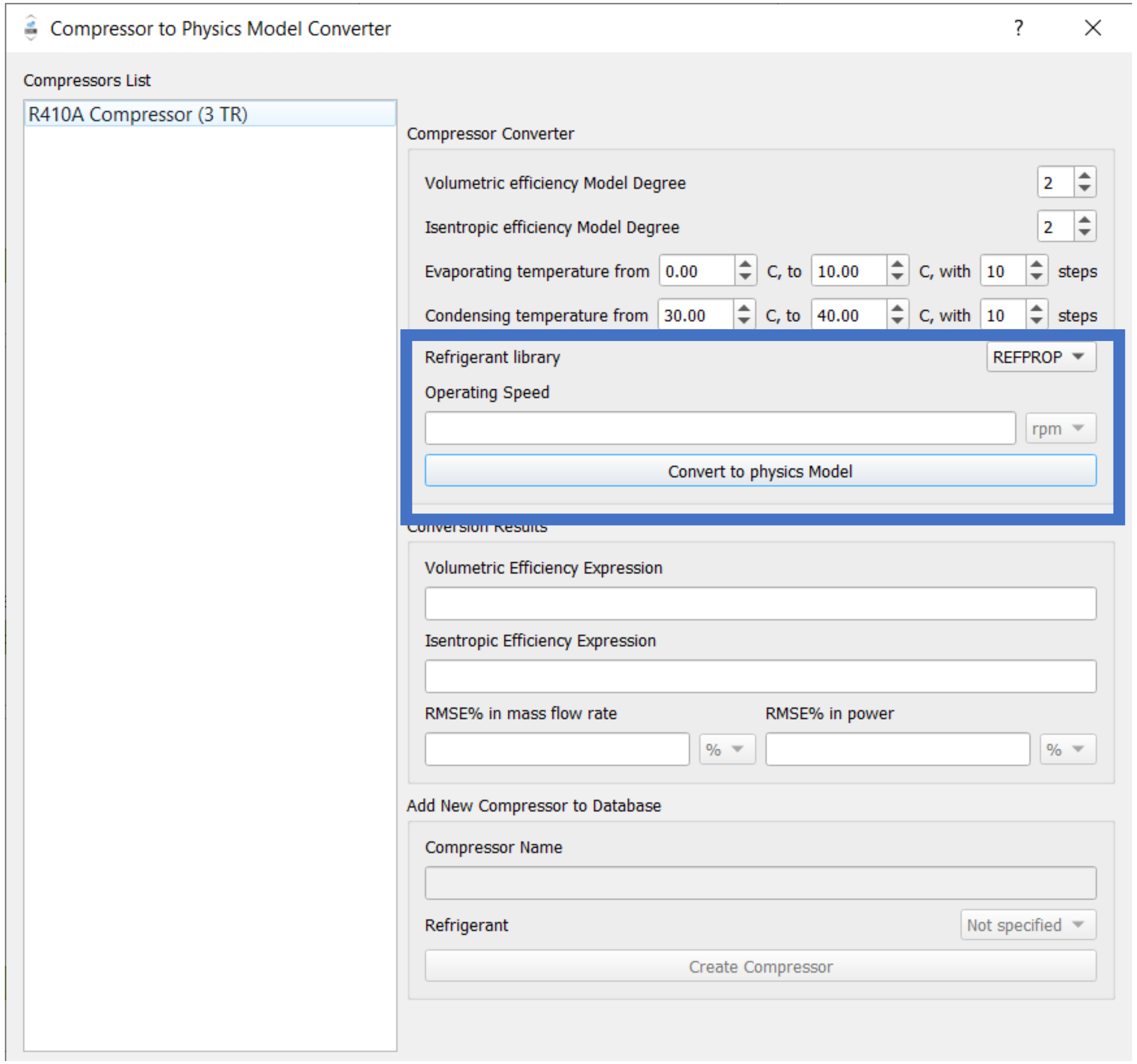
Compressor to Physics Model Converter, cont.

- The user shall choose the number of points used to develop the polynomial functions and the desired range for evaporating and condensing temperatures



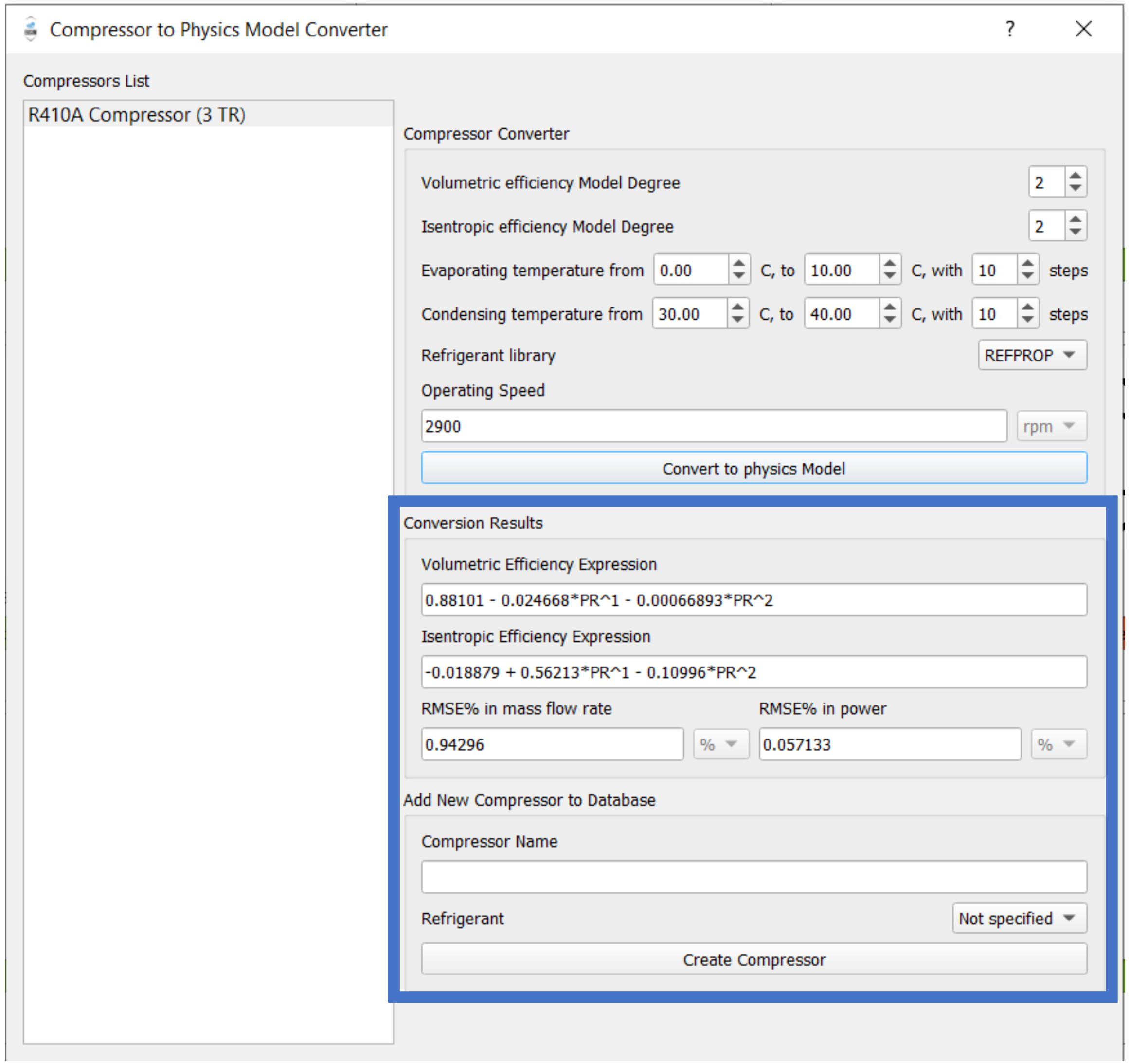
Compressor to Physics Model Converter, cont.

- The user can also choose the preferred refrigerant library:
  - REFPROP
  - HEOS
- The user can also select the compressor operating speed
- Once all inputs are defined, the user should click **Convert to Physics Model** to start conversion



Compressor to Physics Model Converter, cont.

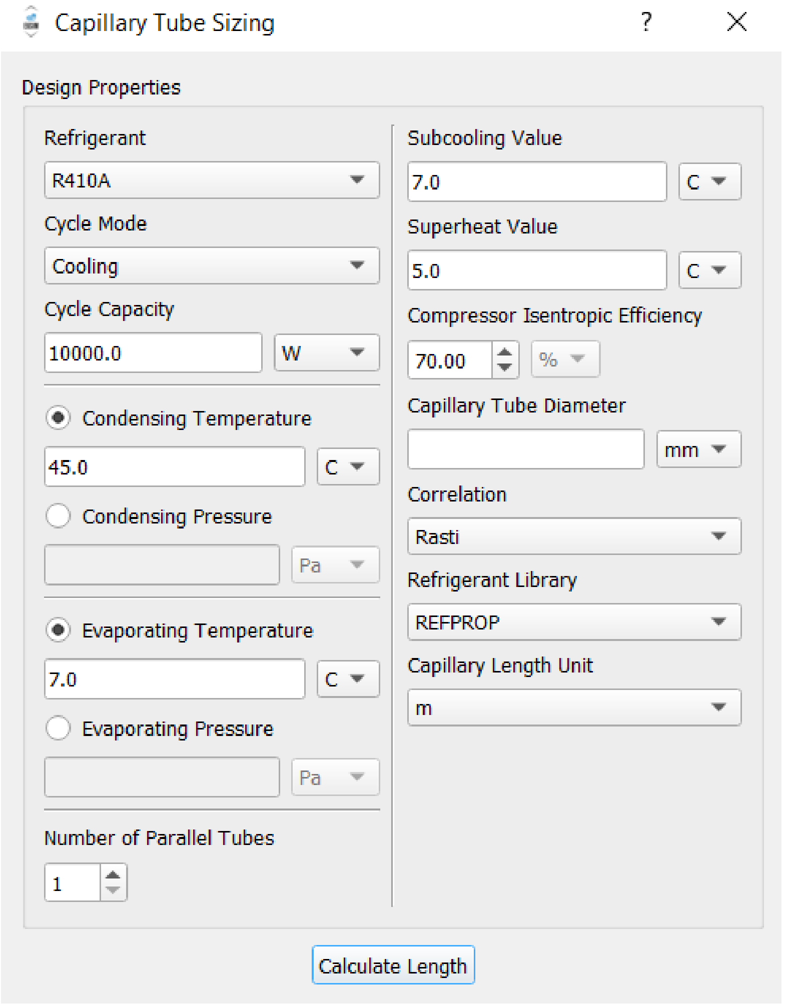
- After conversion is finished, the user will find the conversion results in the **Conversion Results** window which include:
    - The expressions (Polynomials) for the efficiencies
    - Root Mean Squared Error (RMSE) percentage in mass flow rate
    - Root Mean Squared Error (RMSE) percentage in power
- the user can add the converted model to compressor database by entering the **Compressor Name**, the **Refrigerant** (if any), and clicking **Create Compressor**



Capillary Sizing

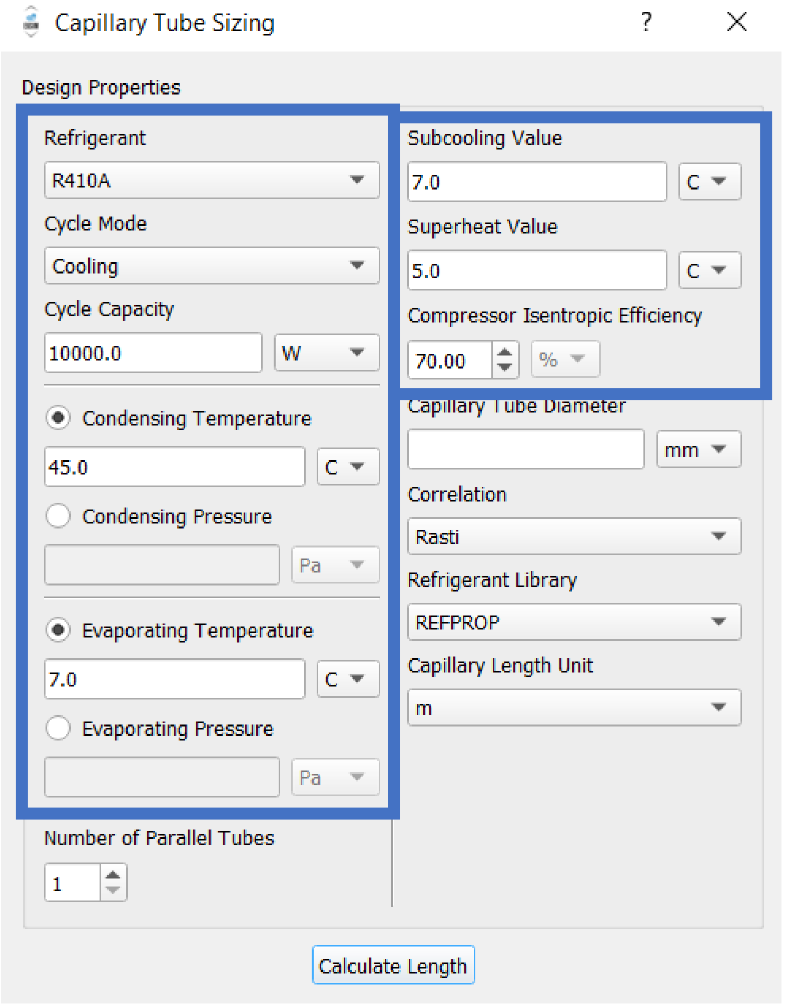
Capillary Sizing

- The **Capillary Tube Sizing** tool is used to find the length of the Capillary Tube based on the main cycle parameters and the Capillary Tube diameter
- The tool will calculate the required refrigerant mass flow rate from the cycle inputs, and size the capillary tube to match the required refrigerant mass flow rate



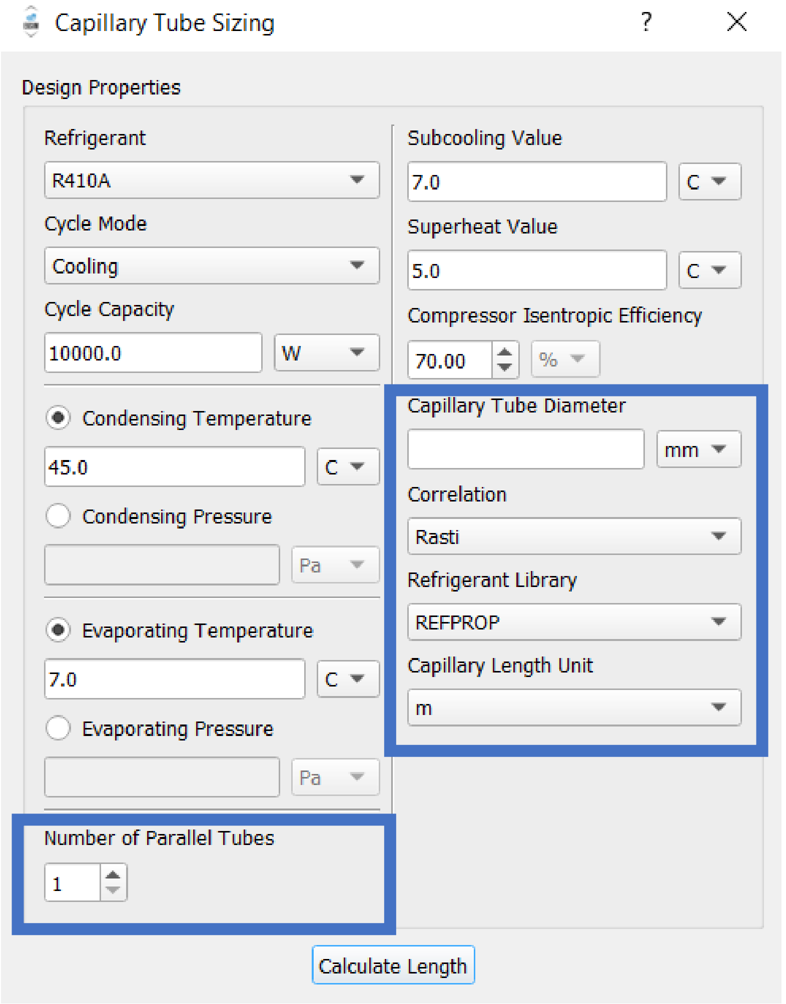
Capillary Sizing, cont.

- First, the user should enter:
  - **Refrigerant**: refrigerant used in the cycle
  - **Mode**:
    - Cooling
    - heating
  - **Capacity**: indoor unit capacity
  - **Condensing temperature** or **Pressure**
  - **Evaporating temperature** or **Pressure**
  - **Subcooling Value**: Condenser Subcooling
  - **Superheat Value**: Evaporator or compressor superheat
  - **Compressor Isentropic Efficiency** *(used only in heating mode)*



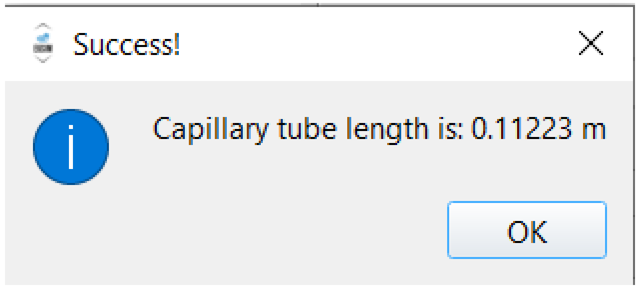
Capillary Sizing, cont.

- Second, the user should enter:
  - **Number of parallel tubes**
  - **Capillary tube diameter**
- Third, choose:
  - **Correlation** to be used in calculating capillary mass flow rate. ***Rasti** is recommended*
  - **Refrigerant library**: library used in calculating refrigerant properties
  - **Capillary length unit**: the unit in which capillary tube length will be displayed
- After finishing all inputs, click **Calculate Length** button



Capillary Sizing, cont.

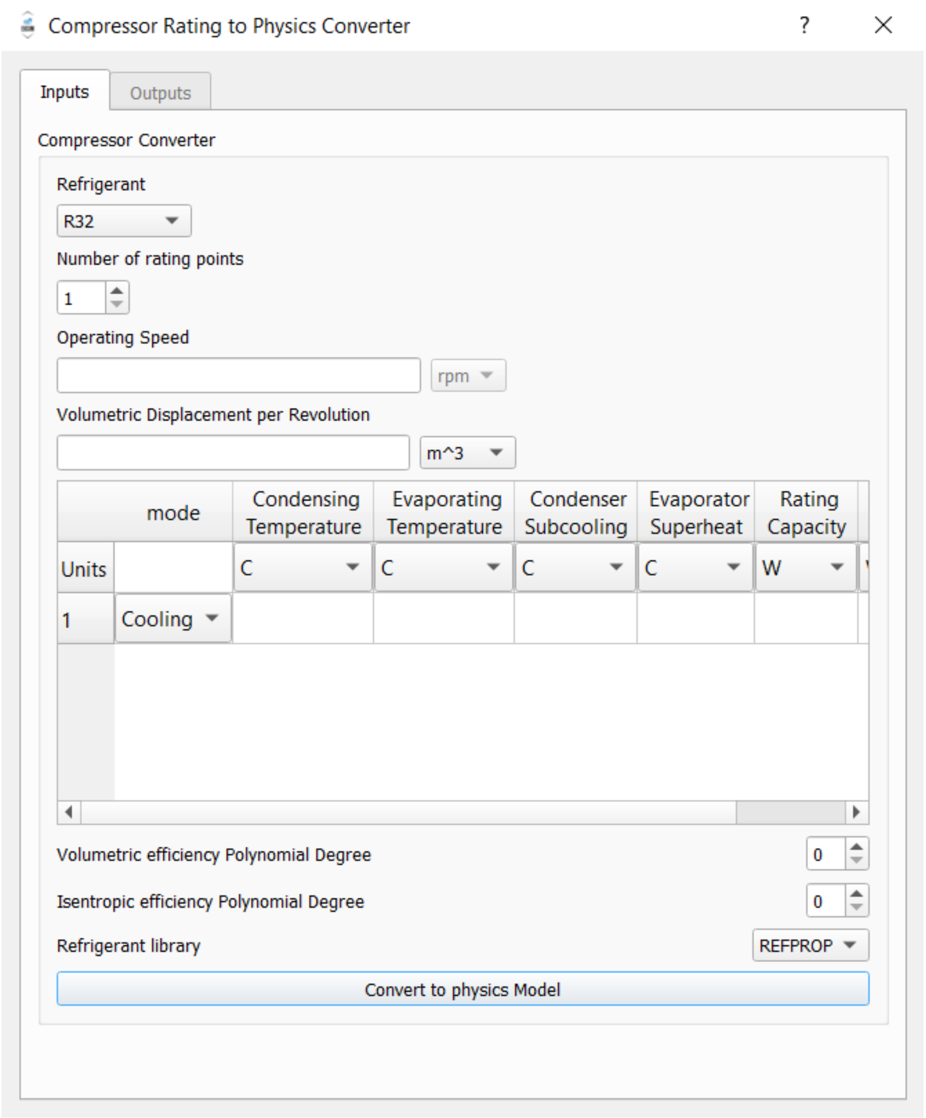
- The tool will show the calculated length in the unit chosen by the user in a dialog box



Compressor Rating to Physics

Compressor Rating to Physics

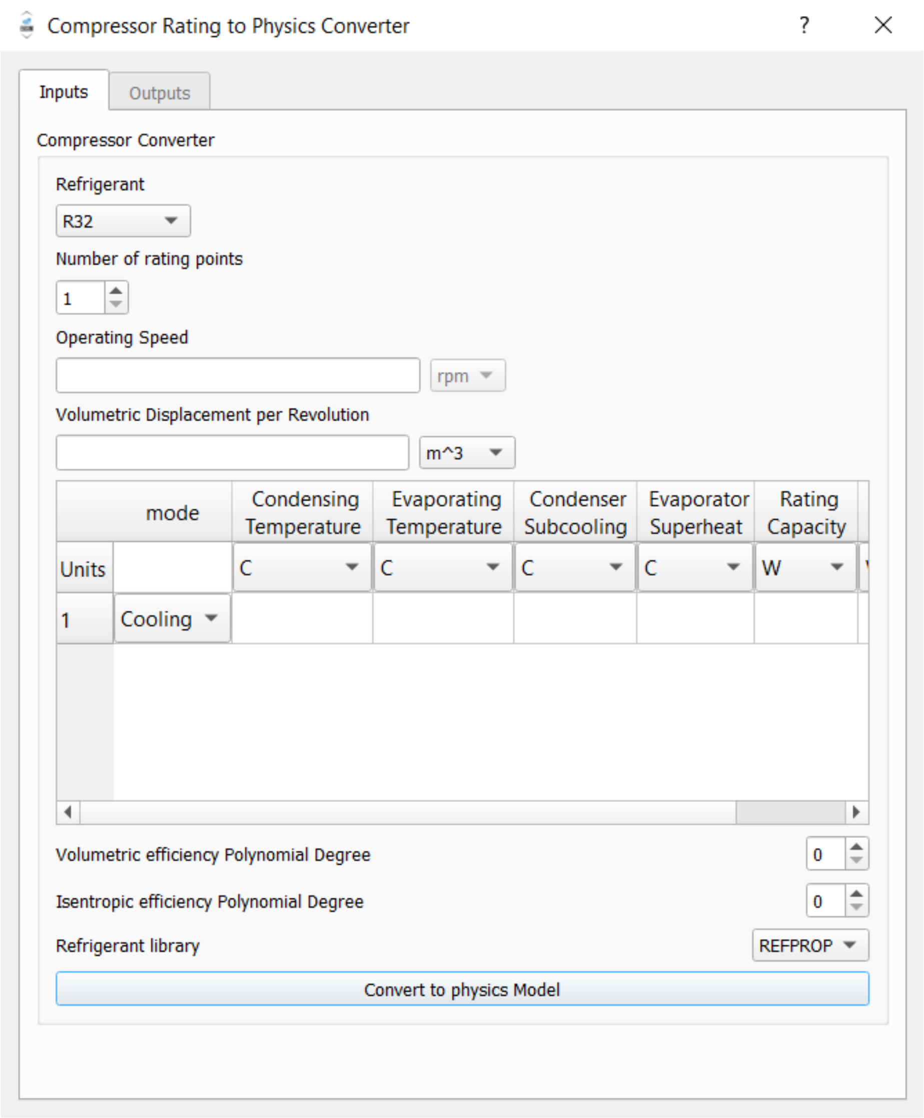
- The tool is used to convert rating point(s) of a compressor to a Physics-based model
- This tool is handy in case the performance map of the compressor is not available, and only one or more rating points are known



Compressor Rating to Physics, cont.

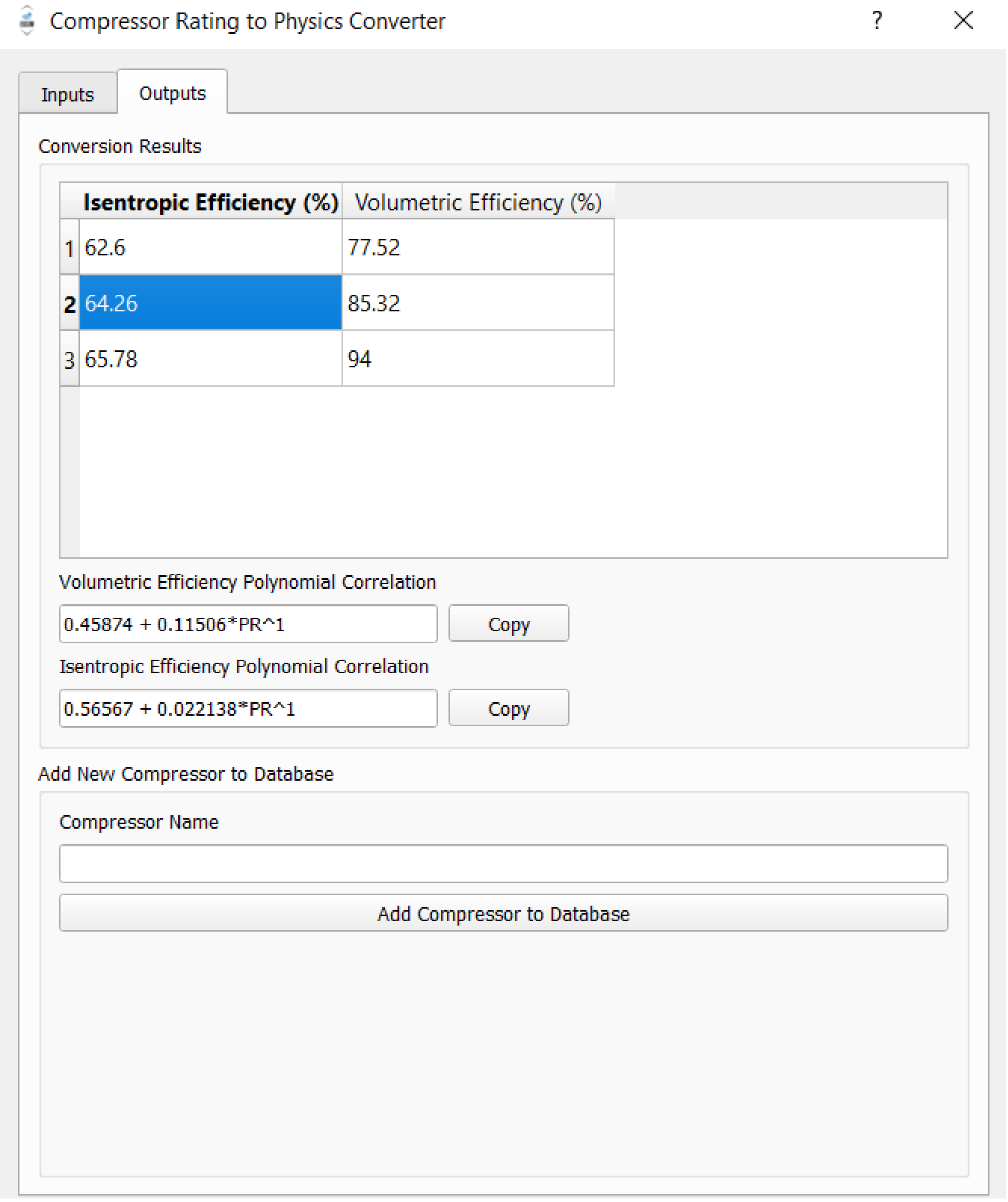
- First, the user has to define:
  - Compressor **Refrigerant**
  - **Number of rating points**
  - Compressor **Operating Speed**
  - Compressor **Volumetric Displacement per Revolution**
- Second, the user has to enter the rating point(s)
- Third, the user should select volumetric and isentropic efficiency polynomial degree.

minimum degree is N-1 where N is the number of rating points, and the recommended degree is N-2



Compressor Rating to Physics, cont.

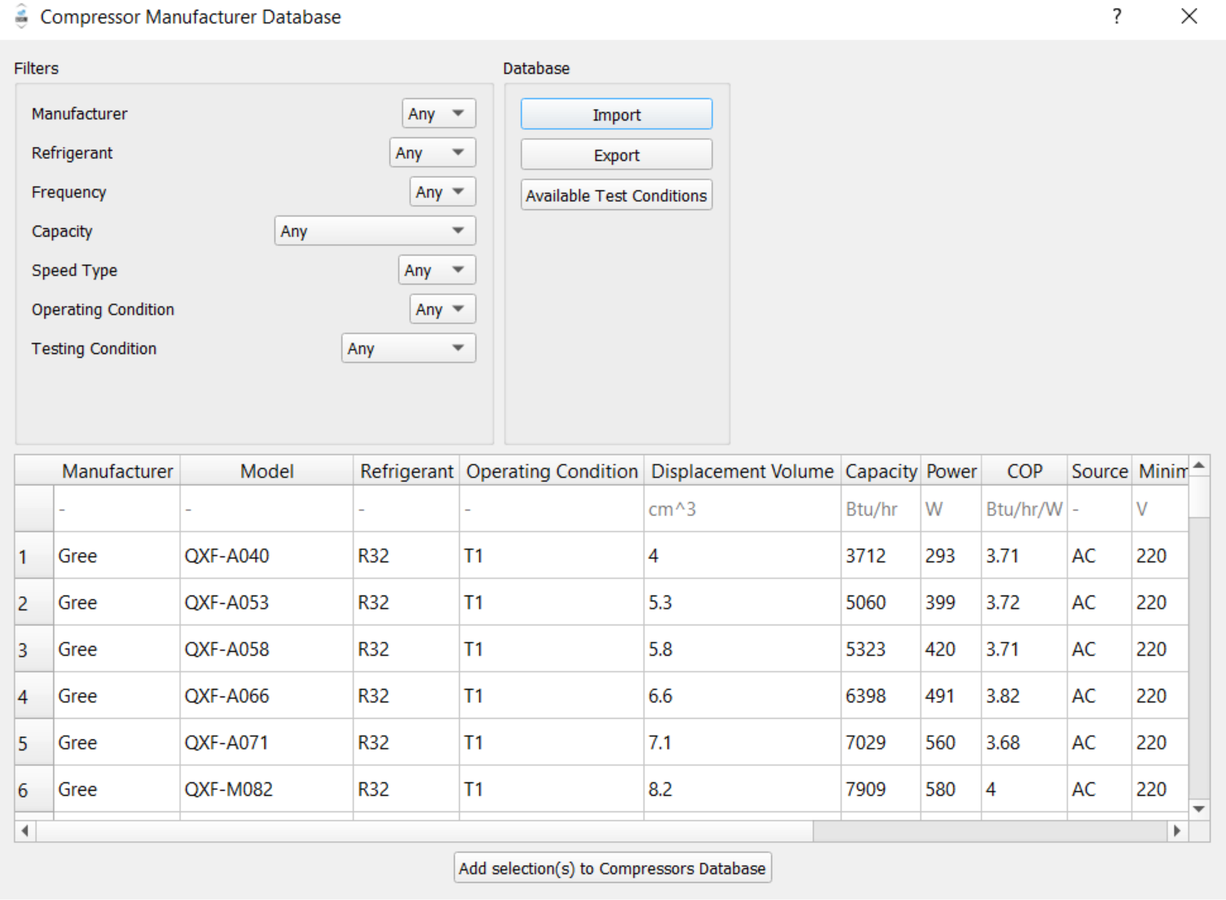
- After conversion, the user can find a new enabled **Outputs** tab with conversion results
- The user can find a table showing the isentropic and volumetric efficiencies for each rating point
- The user can also find the polynomial correlation for isentropic and volumetric efficiencies based on the degree selected
- Finally, the user can **Add Compressor to Database** after choosing a **Compressor Name**



Import Compressor from Manufacturer Datasheet

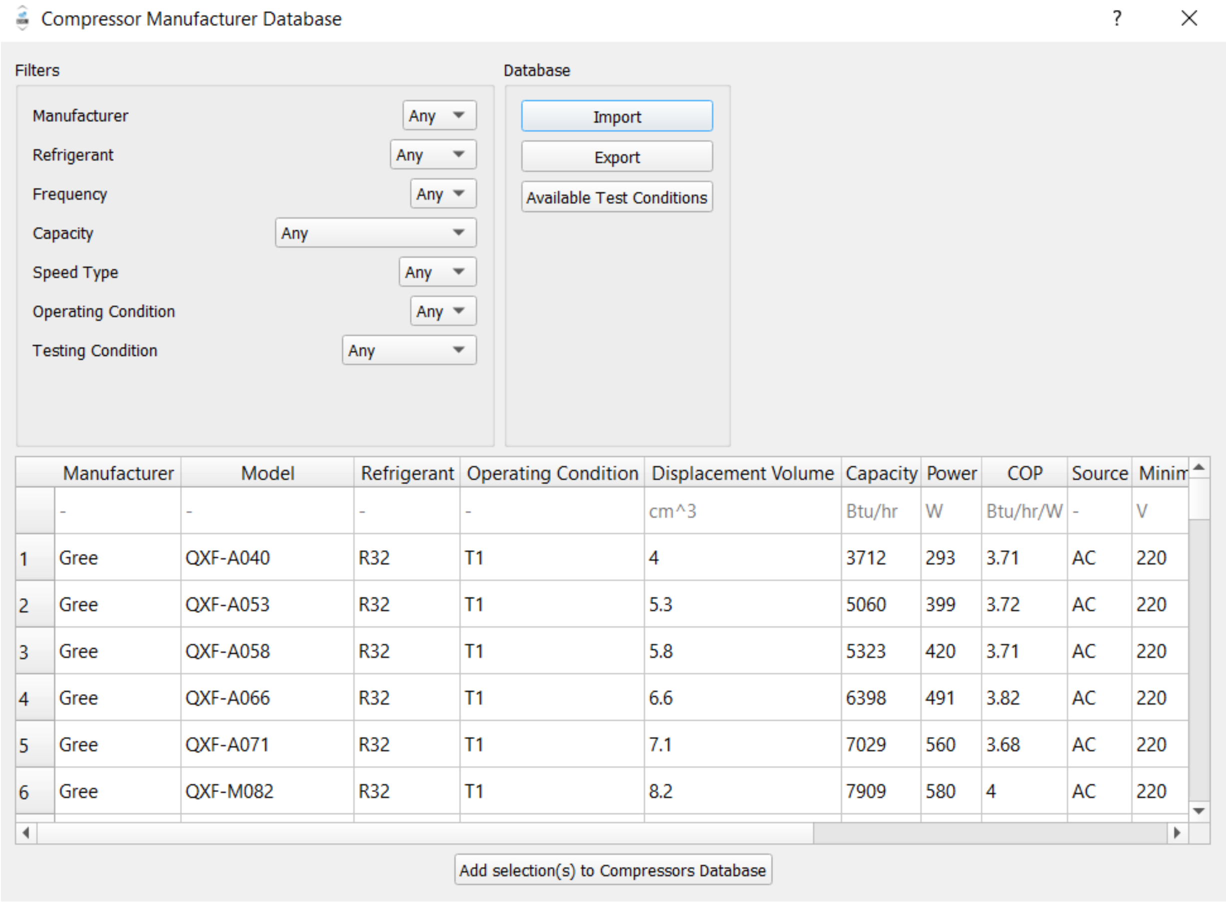
Import Compressor from Manufacturer Datasheet

- EGSim comes with an available list of compressor database from GMCC and GREE
- The user can browse compressors and add a compressor to the Compressor Database
- Compressor information available in manufacturer's datasheet is displayed in the bottom half of the tool
- The user can use relevant **Filters** to limit the number of displayed compressors
- The user can also **Import/Export** a database file, and view the Available Test Conditions at which compressor rating are designated



Import Compressor from Manufacturer Datasheet, cont.

- In case the user wants to add compressor(s) to the EGSim Compressor Database to be used in simulations, he/she can select the compressor(s), then click **Add Selection(s) to Compressor Database**
- The user can find the added Compressor(s) in EGSim Compressor Database

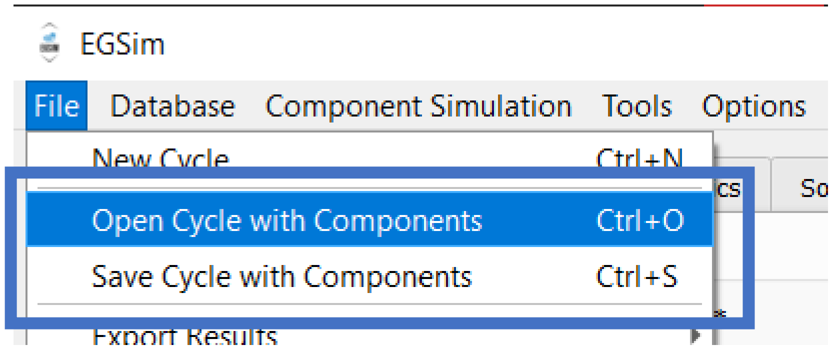


9. miscellaneous

EGSim User Guide

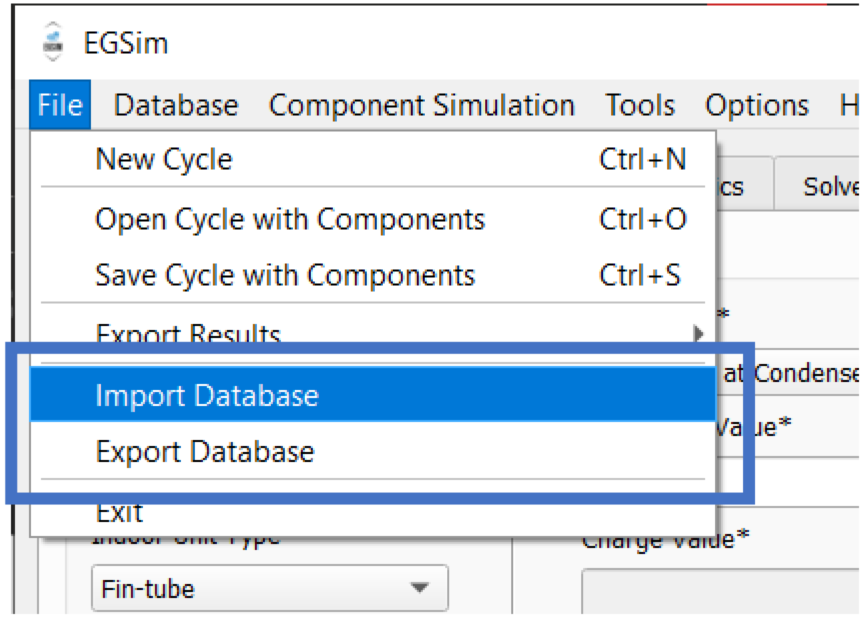
Saving and loading cycle

- To save a Defined Cycle, click
  - **File** => **Save Cycle with Components**
  - This will save the defined cycle parameters, cycle configurations, cycle components, and parametric study table to an external file
- To load a Saved Cycle, click
  - **File** => **OpenCycle with Components**
  - Note that loading a cycle will overwrite current inputs
  - The loaded components will be temporarily added to the relevant components' database until a new cycle is loaded or EGSim is terminated



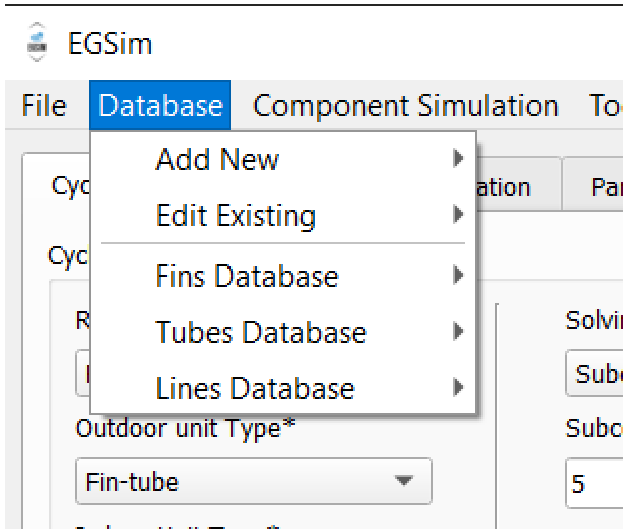
Importing and Exporting Components Database

- To export the current database of the components, click
  - **File** => **Export Database**
  - This will save the current database of components to an external file
- To load a database file, click
  - **File** => **import database**
  - This will remove any current components in the database, and will load all the components from the loaded database file



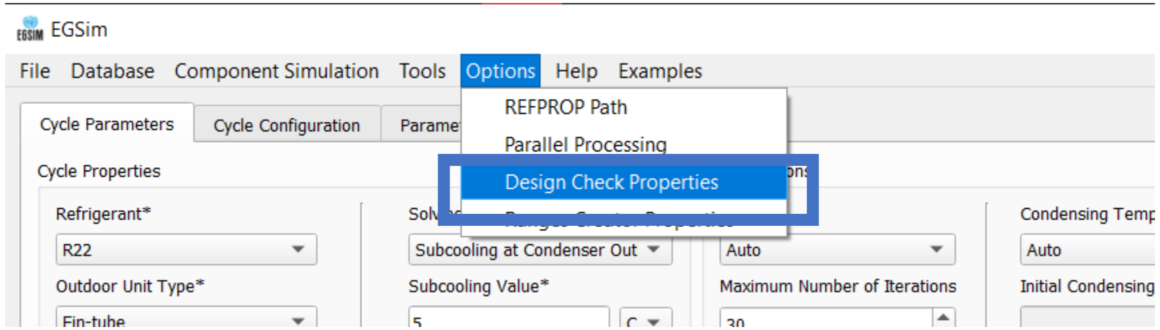
Database

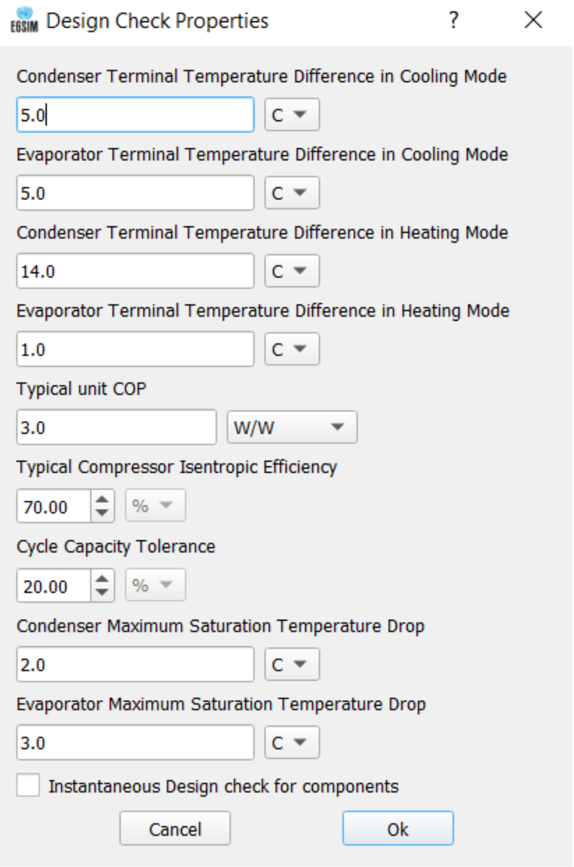
- The user can **Add New** components to the database, or **Edit Existing** components in the database from the **Database** Menu
- The **Database** Menu can also be used to **Edit** or **Import Fins**, **Tubes** or **Line** Databases



Design Check Properties

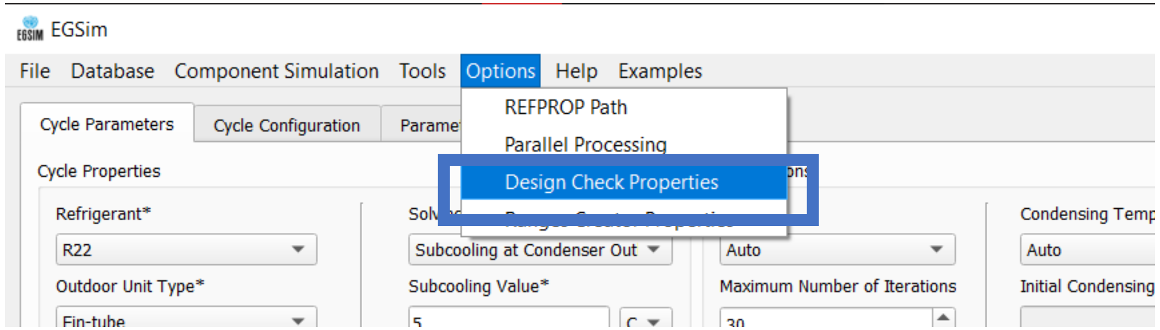
- The user can change the parameters used in **Instantaneous Design Check** and Complete **Design check** from the **Design Check Properties** window
- The properties will be filled with default values defined at first launch of EGSim
- The user can define the **Condenser Terminal Temperature Difference in Cooling and Heating Modes** individually.

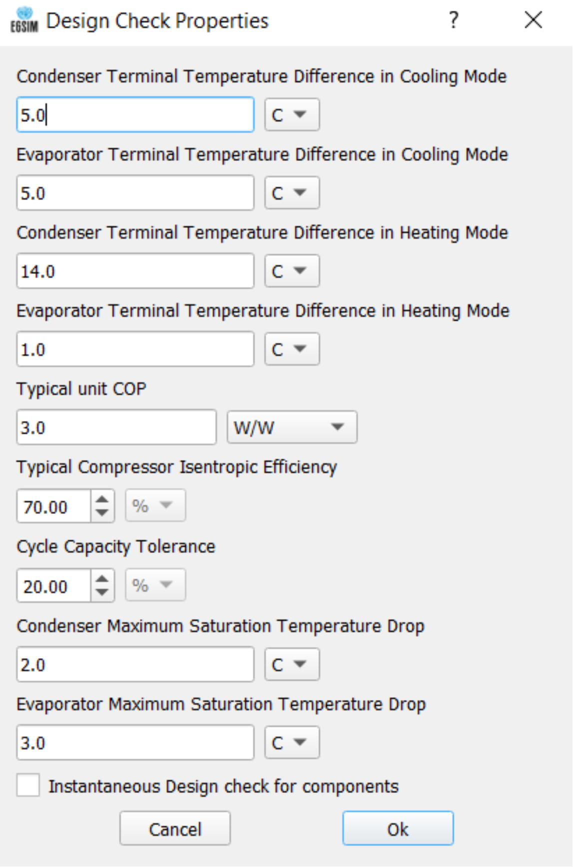




Design Check Properties, cont.

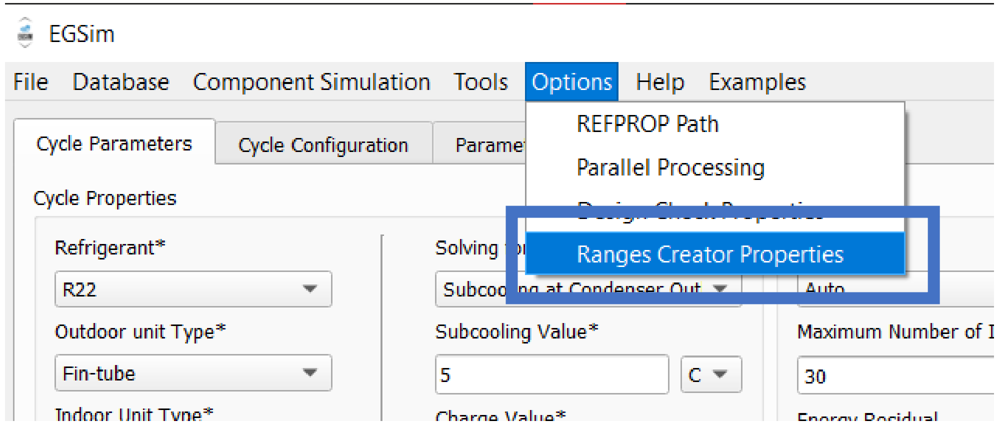
- The user can also define the **Typical unit COP** and **Typical Compressor Isentropic Efficiency** used in calculating condenser or evaporator heat load depending on the cycle mode
- The user can define the criteria for **Cycle Capacity Tolerance** allowance above and below the typical value allowed for the used heat exchangers and the compressor
- The user can define the **Condenser** and **Evaporator maximum saturation Temperature Drop** cause by refrigerant pressure drop in the heat exchanger
- The user can enable/disable the **Instantaneous Design check for components**

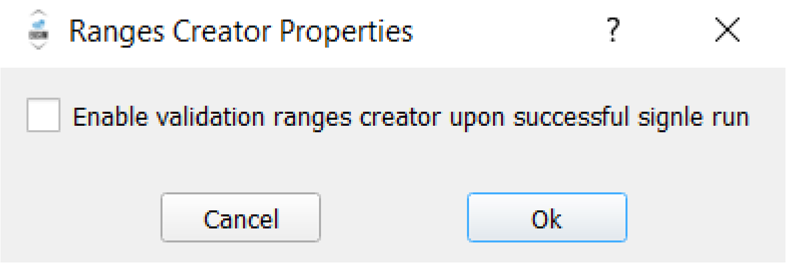




Ranges Creator

- **Ranges Creator** is used to create ranges for different cycle and components' parameters
  - This prevent the user from entering a wrong value that might cause the cycle to fail
  - The Ranges Creator is run after the first successful run and typically takes significant processing time (usually up to 3 minutes depending on the cycle configuration and available processing power)
  - It is advised not to enable the ranges creator during initial design discovery
- To enable the ranges creator, the user should click on the checkbox next to **Enable validation ranges creator upon successful single run**





Examples

- The user can find example cycles from **Examples** Menu
- The user has 3 fully defined examples with different refrigerants and different compressor models