SmartNetics User's Guide 2026.1

by Power Smart Control S.L.

SmartNetics User's Guide

Version 2026.1

Contact and Support

Power Smart Control S.L. is dedicated to assisting and supporting users. For inquiries or assistance, please reach out to Power Smart Control S.L. via email at:

support@powersmartcontrol.com

sales@powersmartcontrol.com

SmartNetics User´s Guide, tutorials or other design advice, services or information, including, but not limited to, reference designs, are intended to assist designers who are developing applications that use SmartNetics; by downloading, accessing or using any particular Power Smart Control resource in any way, you (individually or, if you are acting on behalf of a company, your company) agree to use it solely for this purpose and subject to the terms of this notice.

Power Smart Control reserves the right to make corrections, enhancements, improvements and other changes to its resources.

You understand and agree that you remain responsible for using your independent analysis, evaluation and judgment in designing your applications and that you have full and exclusive responsibility to assure the safety of your applications and compliance of your applications with all applicable regulations, laws and other applicable requirements.

Power Smart Control SL

SmartNetics help

Power Smart Control S.L. makes no representation or warranty with respect to the adequacy or accuracy of this documentation or the software which it describes. Power Smart Control S.L. does not warrant that the software will be uninterrupted or that the operation of the software will be error-free. Power Smart Control S.L. disclaims all representations and warranties, express or implied, as to the software and documentation, including, without limitation, any warranty of merchantability, accuracy, completeness, fitness for any particular purpose, non-infringement, or likelihood of success. In no event will Power Smart Control S.L. or its direct or indirect suppliers be liable for any damages whatsoever including, but not limited to, direct, indirect, incidental, or consequential damages of any character including, without limitation, loss of business profits, data, business information, or any other commercial damages or losses, or for any damages in excess of the list price for the product. In no event shall Power Smart Control S.L. aggregate cumulative liability for any claims arising out of or related to the software exceed the fees or price paid for the software during the preceding twelve-month period. In addition, the following aspects shall apply:

1. Changes and updates: Power Smart Control S.L. is committed to continuous improvement in its products, and, therefore, reserves the right to modify them without prior notification.
2. Accuracy of Information: Despite our efforts to provide accurate information, Power Smart Control S.L. does not guarantee the accuracy or completeness of the information contained in the software and documentation. Users assume the responsibility to verify any critical information regarding the software and documentation for their specific applications.
3. Restrictions on Product Usage and Integration: Unauthorized distribution, replication, or reverse engineering of the software by users is strictly prohibited without explicit written permission from Power Smart Control S.L. Additionally, users are prohibited from incorporating Power Smart Control S.L. software as a constituent part of their own products without obtaining written consent from Power Smart Control S.L.
4. Proper Use: It is the user's responsibility to use the software in accordance with the instructions provided in this document and applicable laws. Power Smart Control S.L. shall not be liable for damages resulting from improper use, negligence, or unauthorized modifications to the software.
5. Liability for Damages/Losses: Power Smart Control S.L. shall not be liable for direct, indirect, incidental, consequential, or special damages/losses (of any nature, without limitation) that may arise from the use of the software, even if the possibility of such damages/losses has been advised.
6. Unauthorized Modifications: Any alterations made to the product, tampering with warranty seals, or suspicion of product modification by users will result in an immediate nullification of the warranty provided by Power Smart Control S.L.
7. Regulatory Compliance: It is the user's responsibility to ensure that the use of the software complies with all applicable regulations and standards in their jurisdiction.

By using this software, the user agrees to the terms and conditions outlined in this clause of disclaimer and limitation of liability, and acknowledges having read and understood the contents of this document.

Introduction

1 Introduction

SmartNetics is a new approach to the design and optimization of magnetic components, aimed at the expert engineer as well as newcomers to magnetic design. It offers a range of features tailored to meet the needs of the electronics industry.

Unlike other commercial software, SmartNetics does not provide a single black-box solution. Instead, it provides the user with every design that meets their needs, allowing them to select the one that is best suited for every particular application, regardless of whether the focus is on performance, volume, cost or any possible combination of parameters.

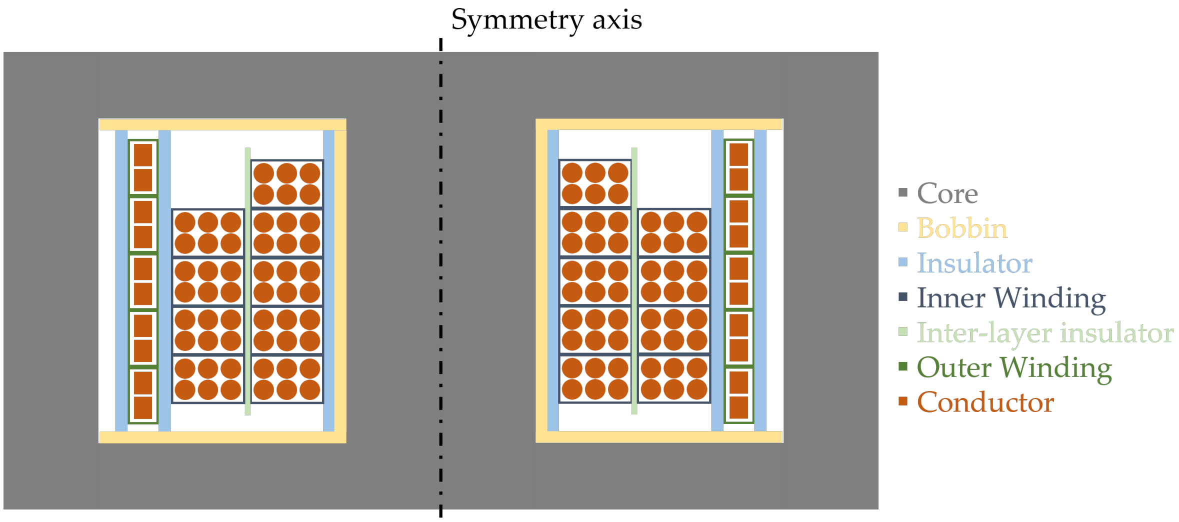
Key Features of SmartNetics:

- **Waveform Input**: Supports sinusoidal, triangular, and rectangular waveforms, as well as custom inputs from .csv files.
- **Geometries**: Includes core configurations with EE shapes, utilizing E or parallel U cores.
- **Winding Strategies**: Offers concentric winding configurations on the center leg.
- **Gapping strategies:** Supports standard gapping (same gap in every leg), single gap in the central leg or distributed gaps in the central leg.
- **Device Types**: Design and analyze single-phase inductors and single-phase transformers with a single secondary.
- **Customizable Databases**: Manage and edit databases from external files.
- **Accurate Thermal Modeling**: Allows dissipation by natural or forced convection.
- **Device Comparison and Selection**: Graphically compare and filter designs across up to eight parameters simultaneously.
- **Integration with Third-Party Tools**: Easily export designs to advanced simulation software like Ansys, Flux or FEMM for finite element analysis and to PSIM for circuit simulation.
- **Professional Report Generation**: Export comprehensive design and analysis reports in PDF format, facilitating documentation and communication with manufacturers.

These help files describe every aspect of the tool. For a guided approach on how to design or analyze a magnetic device using SmartNetics, the user is encouraged to visit our website, which has .pdf and video resources of many of our webinars and tutorials.

1.1 Device parts

The designed devices are composed of several parts, as depicted in the next figure:



The parts shown are:

- **Core**: only EE core shapes are supported in the current version. They can be made out of E semi cores or from U cores in parallel, as described in its corresponding section.
  - The available geometries and materials for the design can be selected in the database tab in configuration.
- **Bobbin**: the coil former can be enabled and configured in its corresponding Bobbin panel inside the device parts tab in configuration. The user can define different thicknesses for the vertical and horizontal sections of the bobbin and select its thermal conductivity for thermal simulations.
- **Insulator**: the insulators are included only if needed. There are three possible insulators: between core and internal winding, between two windings (only for transformers) and between the external winding and the ambient. The insulation requirements can be configured in the Insulators panel inside the device parts tab in configuration.
- **Inner Winding**: for inductors there is only one winding. For transformers this is the inner winding, which corresponds to the one with the highest current. Each turn can be composed by several conductors and is delimited by the blue lines shown in the figure. For a comprehensive description of the paralleling and stacking strategy please refer to the corresponding section.
- **Inter-layer insulator**: the insulators are included only if needed. The inter-layer insulators are added if the voltage difference between adjacent turns requires it. The insulation needs can be configured in the Insulators panel inside the device parts tab in the configuration dialog.

- **Outer Winding**: for inductors the second winding is not present. For transformers this is the outer winding, which corresponds to the one with the lowest current. Each turn can be composed by several conductors and is delimited by the green lines shown in the figure. For a comprehensive description of the paralleling and stacking strategy please refer to the corresponding section.
- **Conductor**: every wire that composes each turn of a winding

1.2 Variables

The variables used in the design procedure for inductors and/or transformers are displayed in the table below. Some variables are only present for inductors while some are only present for transformers.

In the current version, only numerical values can be selected in the drop-down menus inside SmartNetics (text variables, like material or geometry are currently not supported as drop-down selectors). Those variables have a check symbol in the Drop down column in the following table.

Variable	Inductor	Transformer	Drop down	Description
Cores	✓	✓		Geometry of the core
Material	✓	✓		Core material
Stacked	✓	✓	✓	Number of cores horizontally stacked
l_g (m)	✓	✓	✓	Total length of the magnetic flux path through air. If a single gap is used its length corresponds to this one. For designs with more than one gap, this is the summation of their total lengths.
Gaps in legs with winding	✓		✓	Number of gaps in the legs with winding
Gaps in length without winding	✓		✓	Number of gaps in the legs without winding
Stacked cores distance (m)	✓	✓	✓	Horizontal distance between stacked cores
Insulator	✓	✓		Insulator material
Wiring		✓		Defines the winding leg/s around which the winding is made. Currently only central is available

Variable	Inductor	Transformer	Dropdown	Description
Inner winding		✓		Defines which winding is the closest to the central leg. Primary or secondary
N	✓		✓	Number of turns
N_1		✓	✓	Number of turns of the primary winding
N_2		✓	✓	Number of turns of the secondary winding
Window usage	✓	✓	✓	Per-unit portion of the window that is filled by conductors, insulators and bobbin
Primary winding ratio		✓	✓	Horizontal per-unit ratio of the window that is filled by the primary winding
Winding conductor	✓			Name of the wire
Vertical wires	✓		✓	Number of wires in vertical in parallel for every turn of the winding
Horizontal wires	✓		✓	Number of wires in horizontal in parallel for every turn of the winding
Primary wire		✓		Name of the wire for the primary winding
Secondary wire		✓		Name of the wire for the secondary winding
Primary vertical wires		✓	✓	Number of wires in vertical in parallel for every turn of the primary winding
Primary horizontal wires		✓	✓	Number of wires in horizontal in parallel for every turn of the primary winding
Secondary vertical wires		✓	✓	Number of wires in vertical in parallel for every turn of the secondary winding
Secondary horizontal wires		✓	✓	Number of wires in horizontal in parallel for every turn of the secondary winding
Layers	✓		✓	Number of layers of winding
Primary layers		✓	✓	Number of layers of the primary winding
Secondary layers		✓	✓	Number of layers of the secondary winding
BpL	✓		✓	Maximum number of bundles (or turns) in each winding layer

Variable	Inductor	Transformer	Drop down	Description
BpL_1		✓	✓	Maximum number of bundles (or turns) in each primary winding layer
BpL_2		✓	✓	Maximum number of bundles (or turns) in each secondary winding layer
R_{DC}	✓		✓	DC resistance of the winding
R_{DC,1}		✓	✓	DC resistance of the primary winding
R_{DC,2}		✓	✓	DC resistance of the secondary winding
P_{w} (W)	✓		✓	Wire loss
P_{w,1} (W)		✓	✓	Wire loss of the primary winding
P_{w,2} (W)		✓	✓	Wire loss of the secondary winding
P_{c} (W)	✓	✓	✓	Core loss
Winding separation (m)		✓	✓	Empty distance between primary and secondary windings (only if "Max horizontal" winding strategy is used)
Adjusted permeability	✓	✓	✓	Permeability adjusted by current level (only if mu_a is used)
Delta Bpp	✓	✓	✓	Peak-to-peak B field
B_max (T)	✓		✓	Maximum B field
l_w (m)	✓		✓	Wire length
l_w_1 (m)		✓	✓	Wire length of the primary winding
l_w_2 (m)		✓	✓	Wire length of the secondary winding
Total weight (kg)	✓	✓	✓	Total weight of the device
Volume (m^3)	✓	✓	✓	Total volume of the device
Area (m^2)	✓	✓	✓	Total plan area of the device
Loss (W)	✓	✓	✓	Total loss
L_0 (H)	✓		✓	Inductance at 0 current
L_{PT} (H)	✓		✓	Inductance at maximum current (only if mu_a is used)
L_{mu} (H)		✓	✓	Magnetizing inductance referred to primary winding
L_{lk1} (H)		✓	✓	Primary leakage inductance (referred to primary)
L_{lk2} (H)		✓	✓	Secondary leakage inductance (referred to secondary)
Maximum temperature (C)	✓	✓	✓	Maximum temperature at any point of the device
Cost (€)	✓	✓	✓	Total cost of the device

Variable	Inductor	Transformer	Dropdown	Description
Insulator dimensions	✓	✓		Dimensions of every insulator in the design
Temperatures (C)	✓	✓		Temperatures in every part of the device
Weights (kg)	✓	✓		Weight of every part of the device
Prices (€)	✓	✓		Price of every part of the device

1.3 General

Only the devices that comply with every selected option are provided.

If two options contradict each other, the most restrictive is used. Some examples are:

- If a copper wire is fixed for the design but copper is not present and active in the conductor material database, no wire will be used and the design will not produce results.
- If a particular wire geometry is fixed but it does not comply with an imposed restriction, like skin depth, no wire will be used and the design will not produce results.
- If a particular core geometry is fixed but its material is not present and active in the database, no core will be used and the design will not produce results.

1.4 Frequently asked questions

- **I have a particular core geometry and a particular core material selected but there are no designs using that combination.**

Check that said core geometry has that particular material as one of its possibilities in the "Available materials" field in the database.

Remember that the reserved word "Any" can be used to allow any material for that particular geometry.

- **The calculated gap seems to be twice as needed.**

In SmartNetics, the value used for the calculation is not the distance between cores, but the total length of air the flux lines would travel in straight lines.

This relates to gap (distance between cores) differently depending on the gapping strategy:

  - **Standard gapping**: same gap for every leg. The total length of the magnetic path through the air is twice the distance between cores.

  - **Single gap**: a single gap in the central columns. The total length of the magnetic path through the air is the distance between cores.
  - **Distributed gaps**: a number of gaps in the central columns. The total length of the magnetic path through the air is the sum of every gap.

- **Some fields described in this help documentation are not shown in the application.**

  Some fields are only shown when some restrictions are met. Some examples:
  - Voltage waveforms are only present if "Transformer" is selected.
  - Symmetry simplifications in Flux are only available for 3D geometries.

- **There are no available results (or fewer than expected).**

Only the designs that comply with every restriction are presented. Try reducing the restrictions to increase the number of valid results.

- **The maximum number of parallel wires is set to a given number but the designs only get to a lower one.**

The number of wires in parallel is limited by the maximum imposed and by the available window. If the wire does not fit the window using the imposed maximum, a lower number is used.

- **I want a transformer with an air gap.**

In the current version only single-winding devices (inductors) can have an air gap.

- **The measured leakage inductance does not match the predicted one.**

Leakage inductance is closely related to winding positioning and manufacture.

The values provided are an estimation that matches 3D finite element simulations, some differences are expected once the device is manufactured.

- **The PDF generation never ends or does not produce a PDF file.**

The PDF is created by means of a third party software. Please, follow the installation steps shown in the corresponding section.

When installing, please make sure to enable libraries to be installed (automatically or asking), otherwise the PDF cannot be created.

- **Every aspect of the design is set to a certain value, but the design procedure does not produce any result.**

Even if one (or some, or every) design parameter is set (see the corresponding section), only the designs that comply with every imposed criteria are displayed.

If everything is set to a fixed value but that particular design would saturate, or handle a current higher than the maximum, etc., it would not be shown.

Please, reduce the restrictions to ease the consideration of a particular design.

- **Finite element simulations take too long.**

FEM simulations are time and resources consuming, so lengthy simulation times are expected.

To speed up the simulation try avoiding the inclusion of unneeded items for electromagnetic simulations, like coil formers or insulators. Some simplifications can also speed-up the simulation, as described in its corresponding sections for Ansys, Altair or FEMM).

For any question related to SmartNetics, feel free to contact our support by email at: support@powersmartcontrol.com

1.5 Notice

This documentation may contain information about some products (other than SmartNetics) that are developed and under the ownership of external third parties.

ANSYS (Maxwell and Icepak), Altair (Flux), PSIM, FEMM and MikTeX are products developed and under the ownership of different external third parties (compatible with SmartNetics).

The information contained in this document regarding any third-party product is non-official, non-binding, and only for information purposes for the user, based on the best possible understanding, knowledge, and good faith of Power Smart Control S.L.

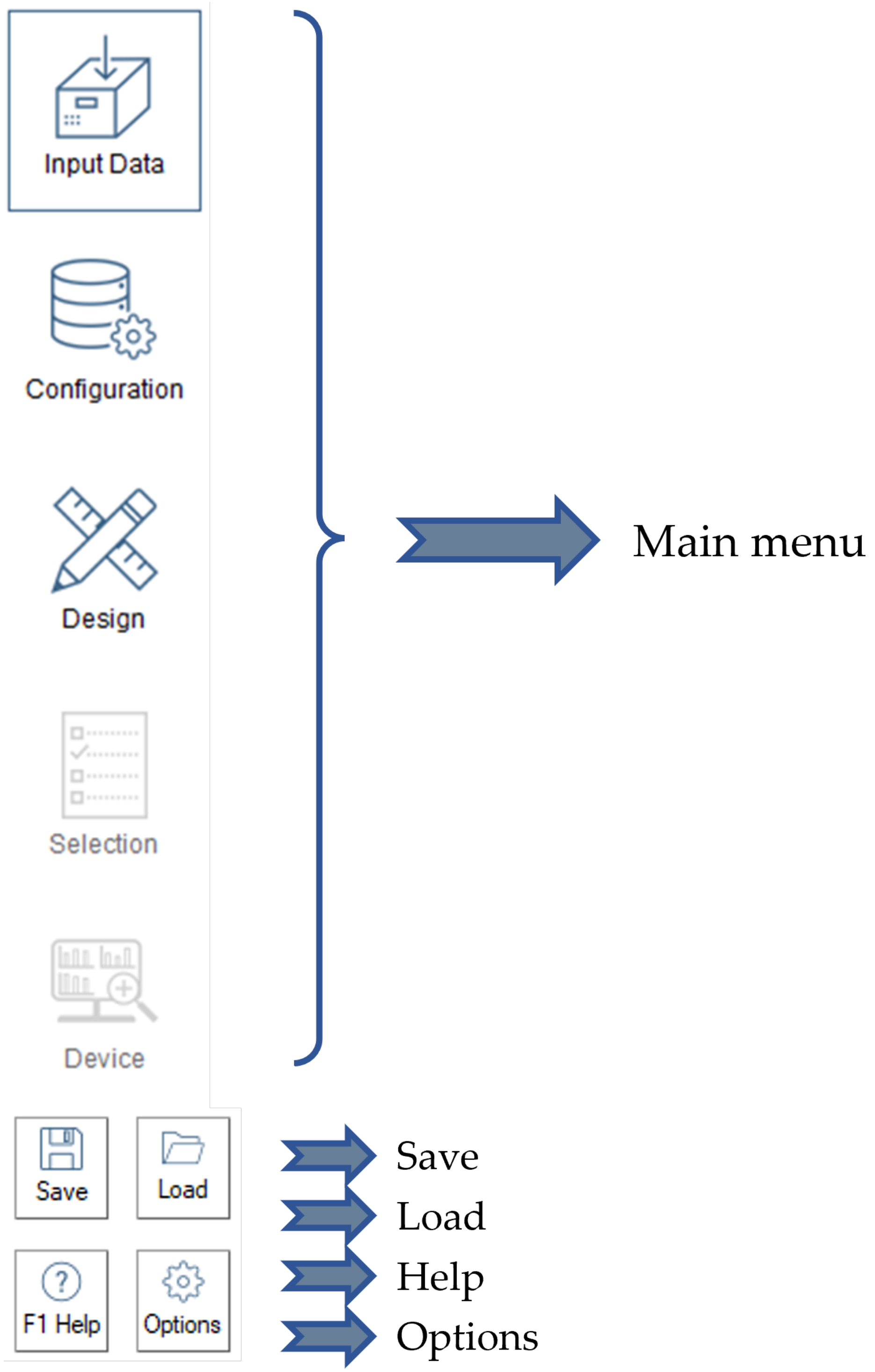
Lateral menus

2 Lateral menus

There are 5 menus, located always at the left side of the window:

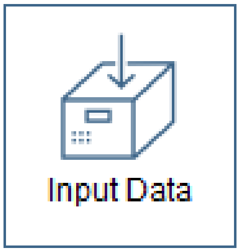
- Main menu
- Save
- Load
- Help
- Options

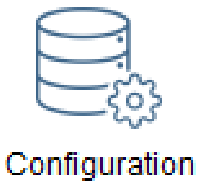
As shown in the figure below:

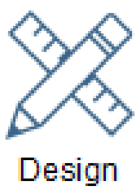


2.1 Main menu

The main menu allows the user to navigate through the different dialogs needed for the design. It consists of 5 buttons, that correspond to the 5 steps used in SmartNetics for the design of a magnetic component:





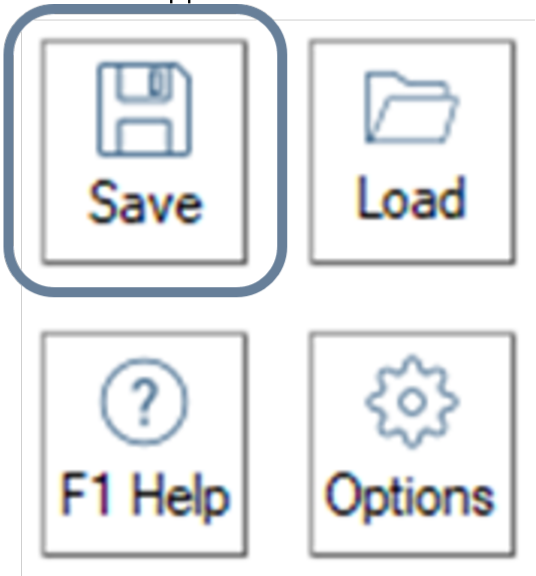


1. **Input data**: Data input dialog. In this dialog the user can define the requirements for the design.
   - Inductance and Current for inductors.
   - Turns ratio and Primary voltage and current for transformers.
2. **Configuration**: Design configuration dialog. In this dialog the user can define the restrictions for the design.
   - Databases to be used.
   - Design limits.

- Models considered for the calculations.
3. **Design**: Begin design dialog. In this dialog the design procedure begins, and results can be analyzed and compared graphically.
   - Here the user can filter out the results to reduce the number of possible designs on the selection step.
   - Generating at least one valid design enables the button for the 4th step (Selection).
4. **Selection**: Device details and selection dialog. In this window the user can access the value of every parameter used in the design and select a particular device for its comprehensive analysis.
   - Selecting a design enables the button for the 5th step (Device).
5. **Device**: Comprehensive analysis dialog. This dialog is divided in several tabs:
   - Geometry. Schematic representation of the full device, core and conductors.
   - Performance. Graphical representation of Temperature, Loss, Weight and Cost distributions.
   - Model impact. Comparison between the results given by the selected models and the available alternatives.
   - Report. Configure and generate a high resolution PDF of every detail.
   - Ansys. Export the selected design to Ansys (Maxwell for electromagnetic simulation and Icepak for thermal).
   - Altair. Export the selected design to Altair-Flux for electromagnetic simulation.
   - FEMM. Export the selected design to FEMM for electromagnetic simulation.
   - PSIM. Export the selected design to PSIM for circuit simulation.

2.2 Save

By means of the Save button, the user can store the design results and the current state of the application.

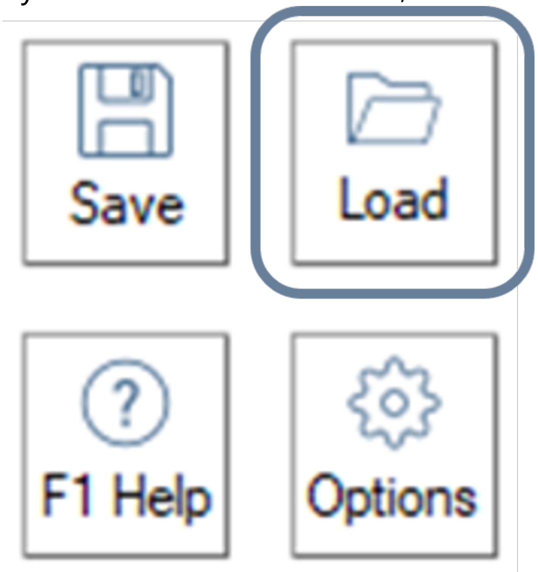


The behavior of the button depends on whether the design procedure has already run or not:

- If there are already results, they are saved along with the configuration used to generate them.
  - If some option has been modified but the design procedure is not run again, that modification will not be stored.
- If there are no results, the current configuration will be saved.
  - The options selected in the moment the button is pushed are the ones saved.

2.3 Load

By means of the Load button, the user can recall a previous state of the application.

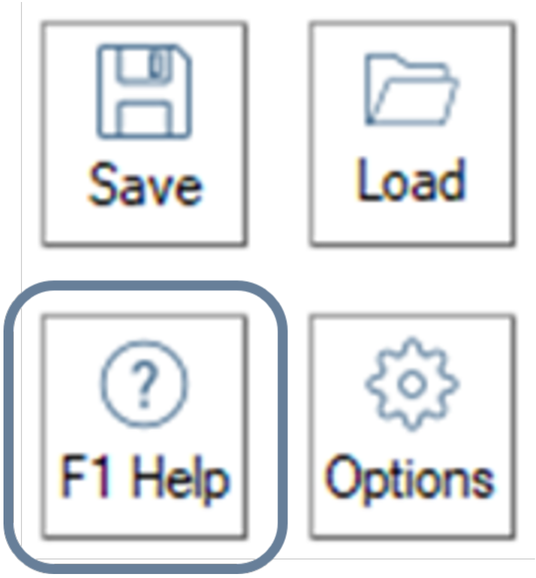


Not only the design results (if any) are loaded, but also the waveforms, options and every application related parameter.

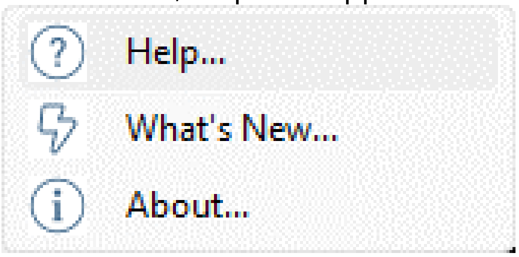
2.4 Help menu

The help menu allows the user to access different topics related to information about the application and the design procedure.

*The user can also press F1 in any part of the application to access the topics related to the current screen.



Once clicked, 3 options appear:

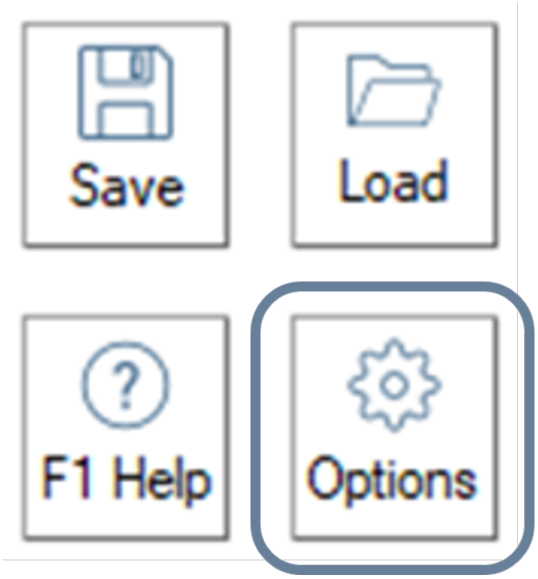


- **Help...**: Opens the help files.
- **What's New...**: Shows the main changes of the current release.
- **About...**: Shows important information about the installed release.

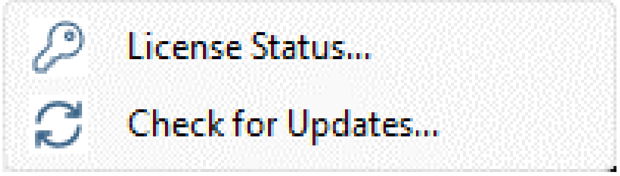
*The contents of the pop-up window may be different for each version and an example is shown below.

2.5 Options

The Options menu shows information about the License and Version of the SmartNetics.



Once clicked, 2 options appear:



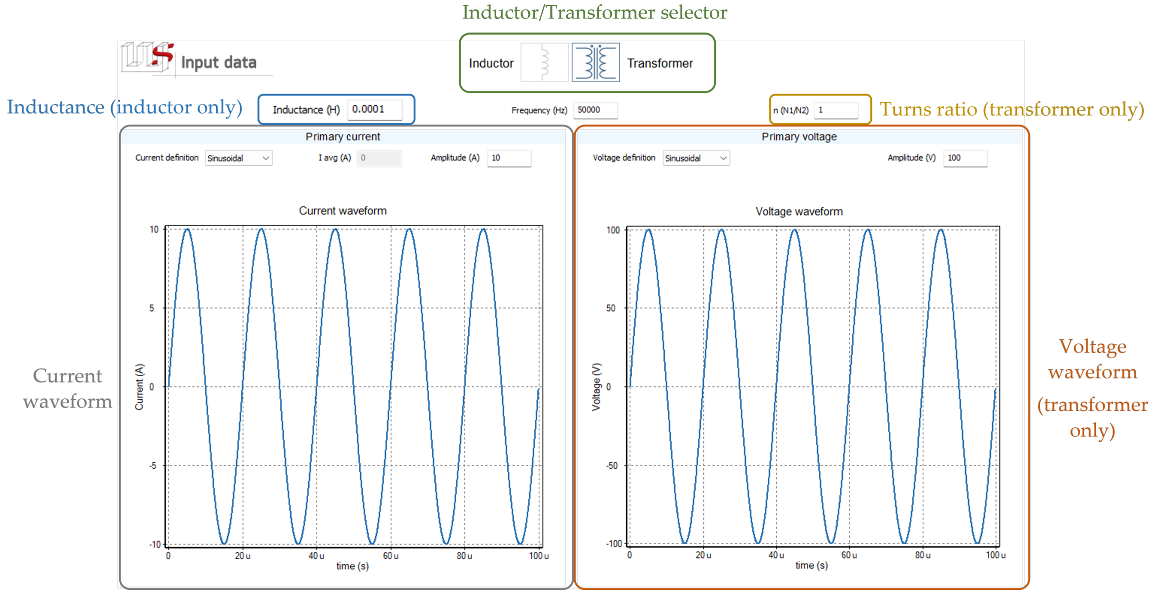
- **License Status...**: Shows a new window with relevant information about the license.

- **Check for Updates...**: Checks if there is an update available.

Input data

3 Input data

In this dialog the user can define the requirements for the design. It is divided in 5 different panels, as shown in the figure below.



Inductor/Transformer selector

This switch allows the selection of the device to design: inductor or transformer. The availability of the rest of the panels depends on the device to be designed:

- **Inductor**: Inductance and current waveform.
- **Transformer**: Turns ratio, current and voltage waveforms.

Inductance (inductor only)

Input the desired inductance value, in Henry.

Turns ratio (transformer only)

Input the desired turns ratio, specified as N1 over N2. N1 is defined as the winding for which voltage and current are provided.

Current waveform

Definition of the current waveform.

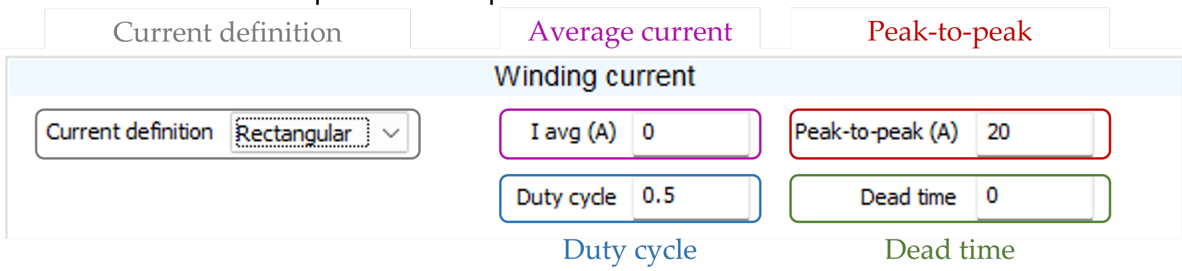
Voltage waveform

Definition of the voltage waveform.

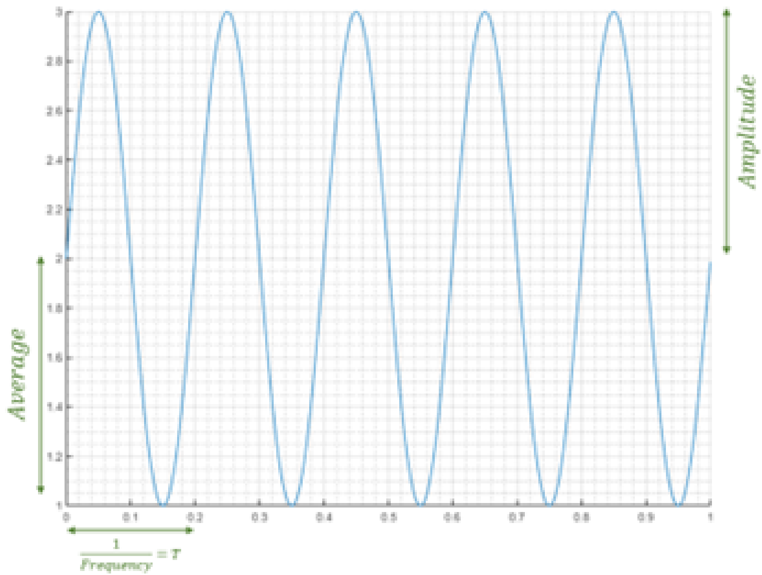
3.1 Current waveform definition

Current waveform

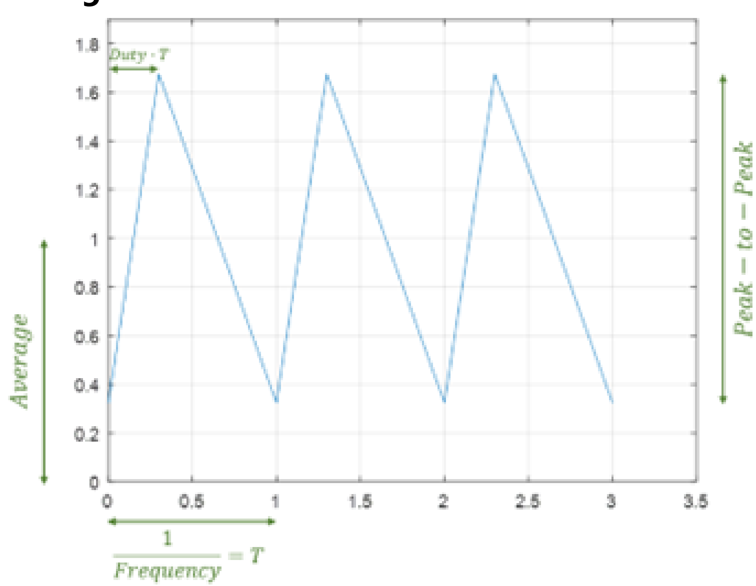
The current waveform panel is composed of different selectors:



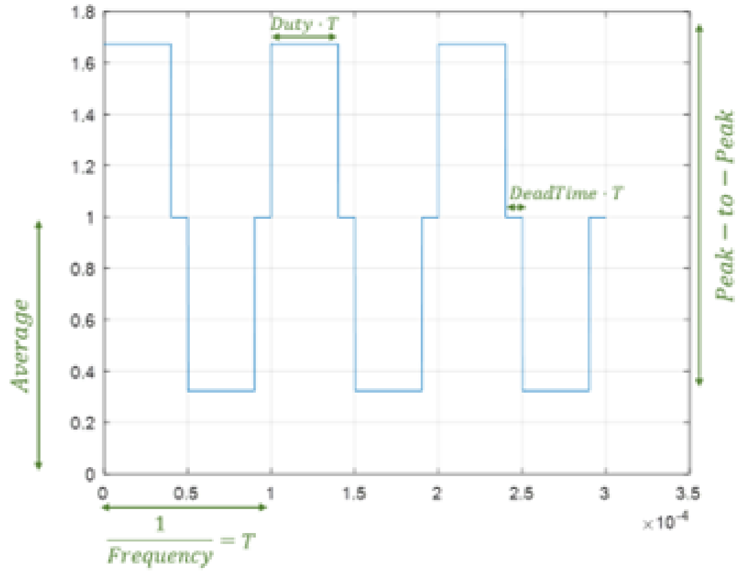
- **Current definition**: Select how the waveform is defined. The current options are:
  - **Sinusoidal**



  - **Triangular**



- **Rectangular**



- **From file**. A dialog opens to select a .csv file with the current waveform.
  - The file must have a .csv extension and be composed of two columns of comma separated values. First column must be times (with any column name) and second must be current values (with any column name), for example:

```
 Time                Current
1.989995000E-02,-3.535595755E+02
1.990000000E-02,-3.535602873E+02
1.990005000E-02,-3.535609826E+02
1.990010000E-02,-3.535616614E+02
1.990015000E-02,-3.535623238E+02
1.990020000E-02,-3.535629698E+02
1.990025000E-02,-3.535635993E+02
1.990030000E-02,-3.535642125E+02
1.990035000E-02,-3.535648092E+02
1.990040000E-02,-3.535653895E+02
1.990045000E-02,-3.535659534E+02
1.990050000E-02,-3.535665010E+02
1.990055000E-02,-3.535670322E+02
1.990060000E-02,-3.535675471E+02
```

The remaining available fields depend on the selected waveform:

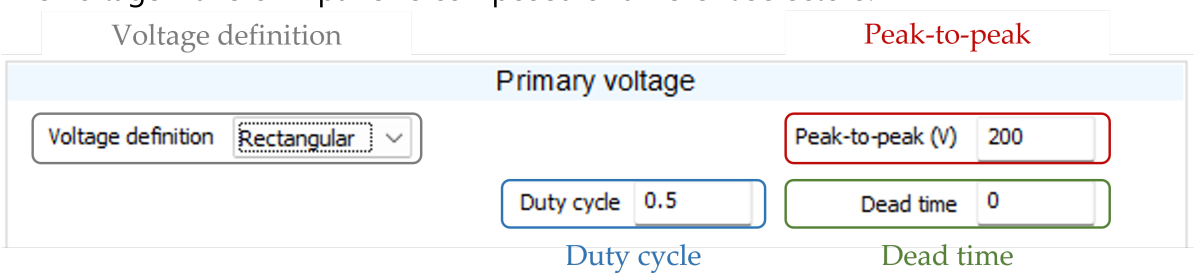
- **Average current**: Average value of the current waveform, in Amps.
- **Peak-to-peak** (Available for triangular and rectangular): Minimum to maximum value of the waveform, in Amps.
- **Amplitude** (Only for sinusoidal): Average to maximum value of the waveform, in Amps.

- **Duty cycle** (Available for triangular and rectangular): Amount of time the waveform rises or stays at its maximum value. Defined as per-unit of the period (0: no dead time, 1: full dead time).
- **Dead time** (Only for rectangular): Amount of time the waveform stays on its average value (after staying at the maximum level or after the rising portion). Defined as per-unit of the period (0: no dead time, 1: full dead time). A single value is defined but applies to both the time after on-time and after off-time.
- **Load file** (Only for file definition): Opens the file browser to search for a current waveform file.

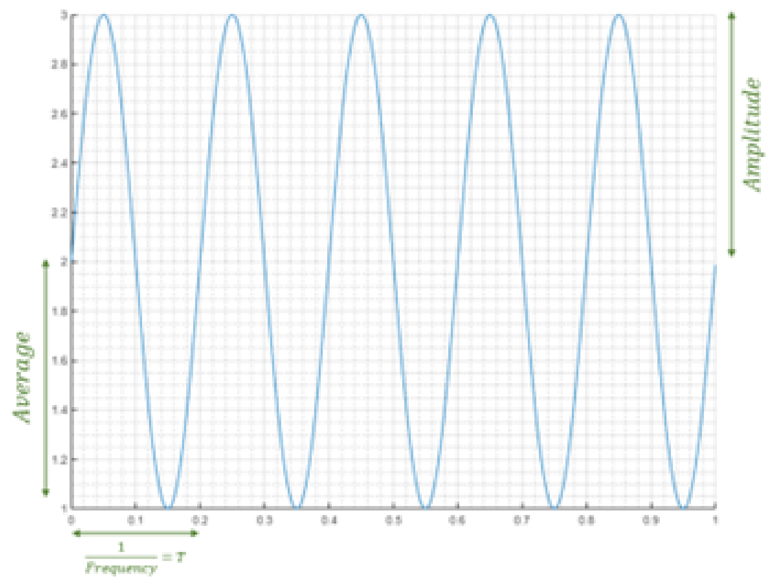
3.2 Voltage waveform definition

Voltage waveform

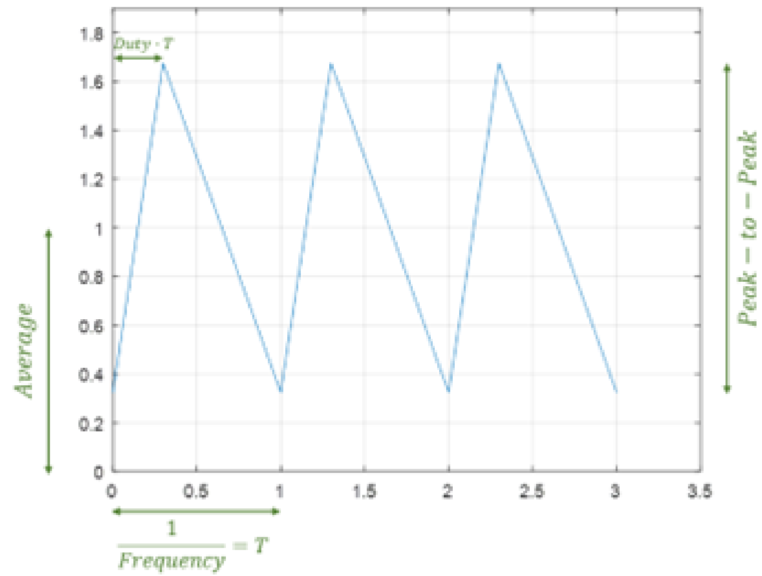
The voltage waveform panel is composed of different selectors:



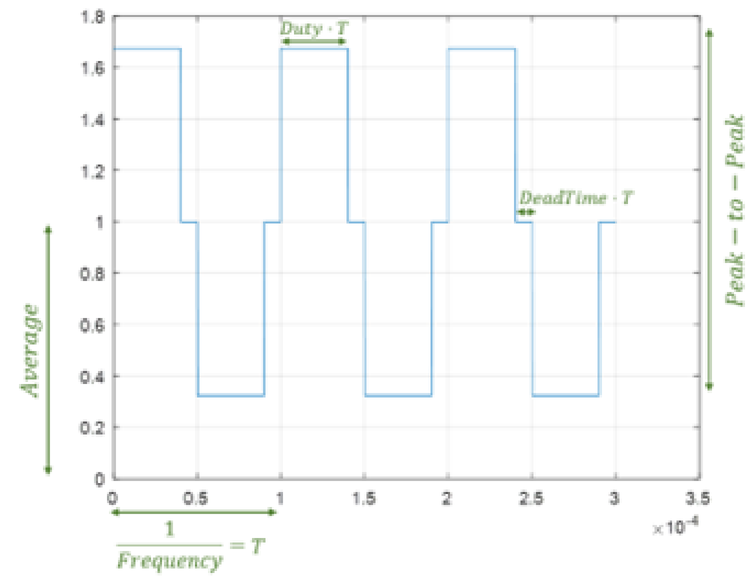
- **Voltage definition**: Select how the waveform is defined. The current options are:
  - **Sinusoidal**



- **Triangular**



- **Rectangular**



- **From file**. A dialog opens to select a .csv file with the voltage waveform.
  - The file must have a .csv extension and be composed of two columns of comma separated values. First column must be times (with any column name) and second must be voltage values (with any column name), for example:

```
Time              Voltage
1.989995000E-02,-3.535595755E+02
1.990000000E-02,-3.535602873E+02
1.990005000E-02,-3.535609826E+02
1.990010000E-02,-3.535616614E+02
1.990015000E-02,-3.535623238E+02
1.990020000E-02,-3.535629698E+02
1.990025000E-02,-3.535635993E+02
1.990030000E-02,-3.535642125E+02
1.990035000E-02,-3.535648092E+02
1.990040000E-02,-3.535653895E+02
1.990045000E-02,-3.535659534E+02
1.990050000E-02,-3.535665010E+02
```

The remaining available fields depend on the selected waveform:

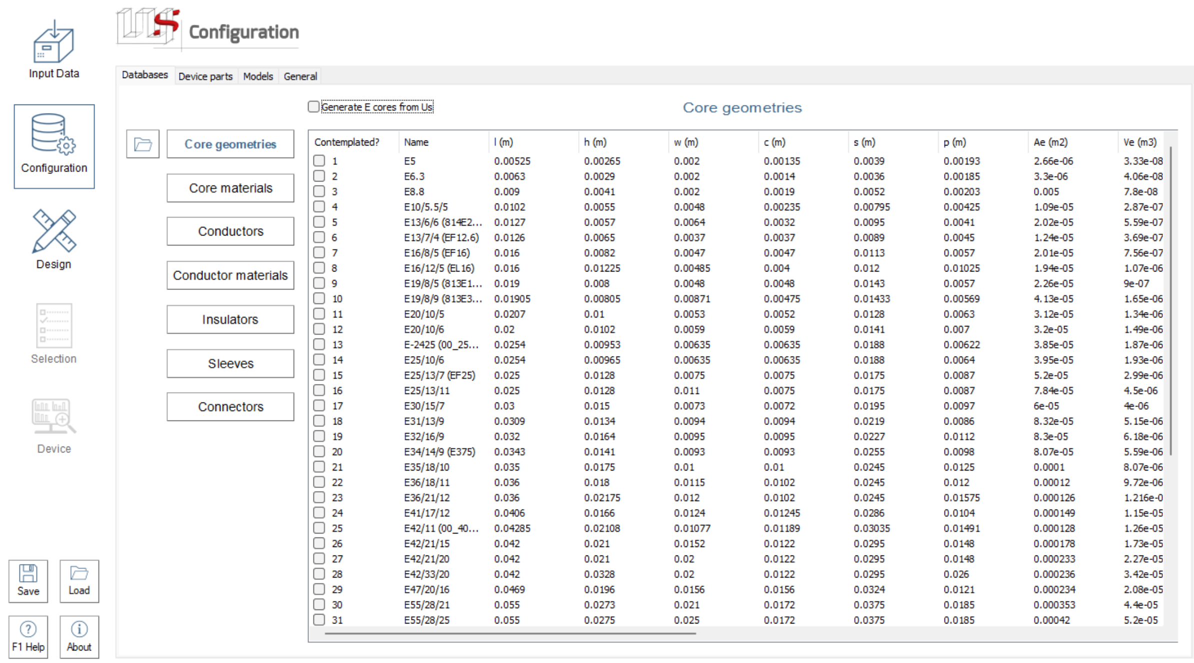
- **Peak-to-peak** (Available for triangular and rectangular): Minimum to maximum value of the waveform.
- **Amplitude** (Only for sinusoidal): Average to maximum value of the waveform.
- **Duty cycle** (Available for triangular and rectangular): Amount of time the waveform rises or stays at its maximum value. Defined as per-unit of the period (0: no dead time, 1: full dead time).
- **Dead time** (Only for rectangular): Amount of time the waveform stays on its average value (after staying at the maximum level or after the rising portion). Defined as per-unit of the period (0: no dead time, 1: full dead time). A single value is defined but applies to both the time after on-time and after off-time.
- **Load file** (Only for file definition): Opens the file browser to search for a voltage waveform file.

Configuration

4 Configuration

This dialog allows the user to configure the design procedure, from databases to restrictions, including model selection, setting of parameters or accuracy needs.

The user can navigate the 4 available tabs by means of the tab selector.



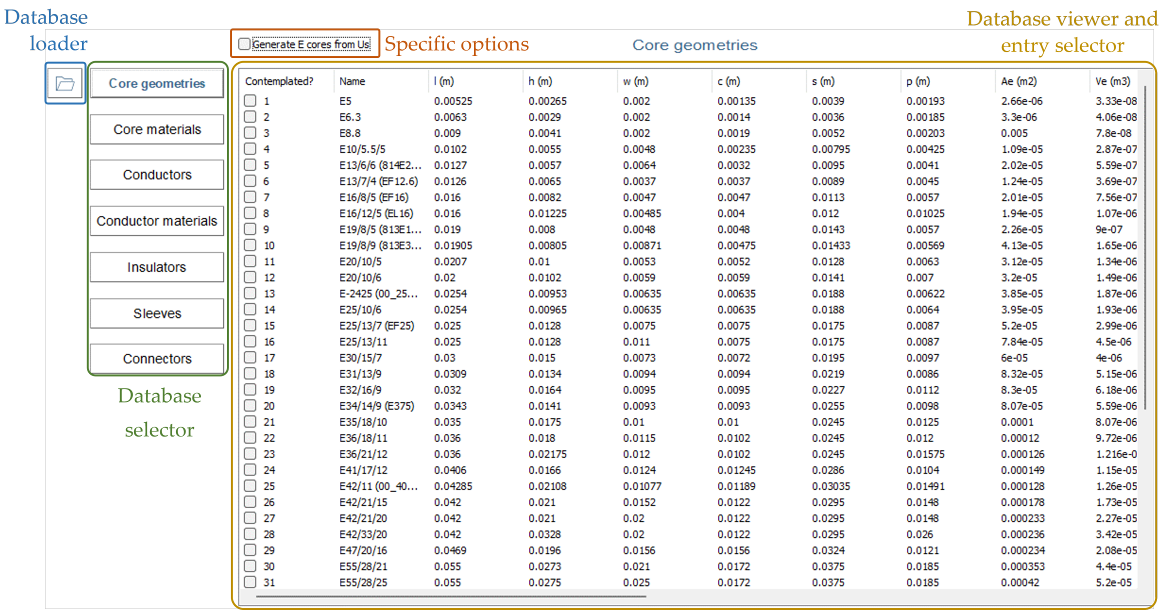
The panel is divided in 4 tabs:

- Databases: In this tab the user can configure what components to consider for the design, regarding:
  - Core geometries.
  - Core materials.
  - Conductors (Conductor geometries).
  - Conductor materials.
  - Insulators.
  - Sleeves.
  - Connectors.
- Device parts: In this tab the user can select the restrictions to impose to every part of the device:
- Models: In this tab the user can select which models to use for the different calculations (inductance, losses and temperature).
- General: In this tab the user can set different parameters to a given value, to restrict the design possibilities.

4.1 Databases

This tab allows the user to access every parameter related to databases loading and manipulation.

It is divided in 4 panels:



- **Database loader**: Allows the user to re-load any database. The currently available databases are:
  - o Core geometries
  - o Core materials
  - o Conductors
  - o Conductor materials
  - o Insulators
  - o Sleeves
  - o Connectors

- **Database selector**: Select which database to display.

- **Database viewer and filter**: Shows every entry of the currently selected database and allow the user to select whether or not to use a particular entry.

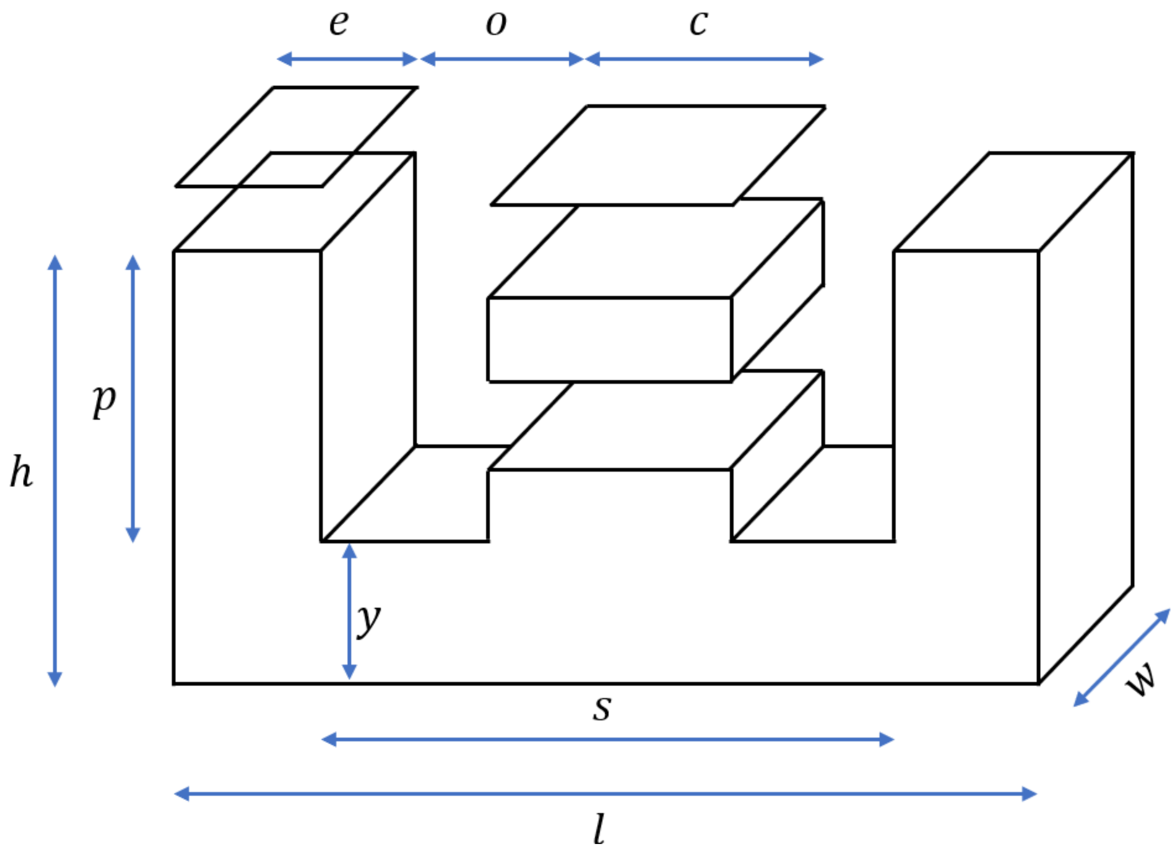
- **Specific options:** This options depend on the particular database selected and can be:
  - o Generate E cores from Us: Select whether to use 2 U cores in parallel to make E cores or not, as described in its corresponding section.
  - o Force same conductor (only for transformers): Force every design to use the same conductor for primary and secondary windings.

4.1.1 Core geometries

The core geometry database is composed of the following parameters (see reference image for a graphical description):

- **Contemplated?**: choose if this particular geometry is to be considered for the design.
- **Name**: name of the core geometry.
- **l (m)**: **l**ength of the core, in meters.
- **h (m)**: **h**eight of the core half, in meters.
- **w (m)**: **w**idth of the core, in meters.
- **c (m)**: **c**entral leg thickness, in meters.
- **s (m)**: **s**eparation between external legs, in meters.
- **p (m)**: **p**illar height, in meters.
- **Ae (m2)**: effective cross-sectional area of a core, in square meters.
- **Ve (m3)**: total effective volume of the core (assuming two equal core halves), in cubic meters.
- **lm (m)**: total effective length of the core (assuming two equal core halves), in meters.
- **o (m)**: **o**pening, distance between adjacent legs, in meters.
- **e (m)**: **e**xternal leg thickness, in meters.
- **y (m)**: **y**oke height, in meters.
- **Predefined gaps (m)**: fixed gap lengths (vertical distance between two core parts), in meters. Only used if the "Force predefined gaps" option is selected in Device parts.
- **Gaps per semicore**: fixed numbers of gaps for each core half. A number higher than one generates distributed gaps if the "Allow distributed gaps" option is active. Only used if the "Force predefined gaps" option is selected in Device parts.
  - If this value is set to 1, only a single gap in the central column will be used. If it is set to a different value, the total number of gaps is twice that value. For example, a value of 2.5 implies 5 gaps in total in the central column.
- **A_L (H/N^2)**: area product of the core (assuming two equal core halves).
- **Available materials**: materials in which the geometry is available.
  - **IMPORTANT NOTICE:** To allow a particular geometry to be used with any material, the reserved word "Any" can be used. That way, any core material active in the database is considered for that particular geometry.
- **Prices (€)**: price of a core half.
- **Manufacturer**: manufacturer of the part (this field has no impact on the design).
- **Geometry**: geometry of the part (E or U are currently available).

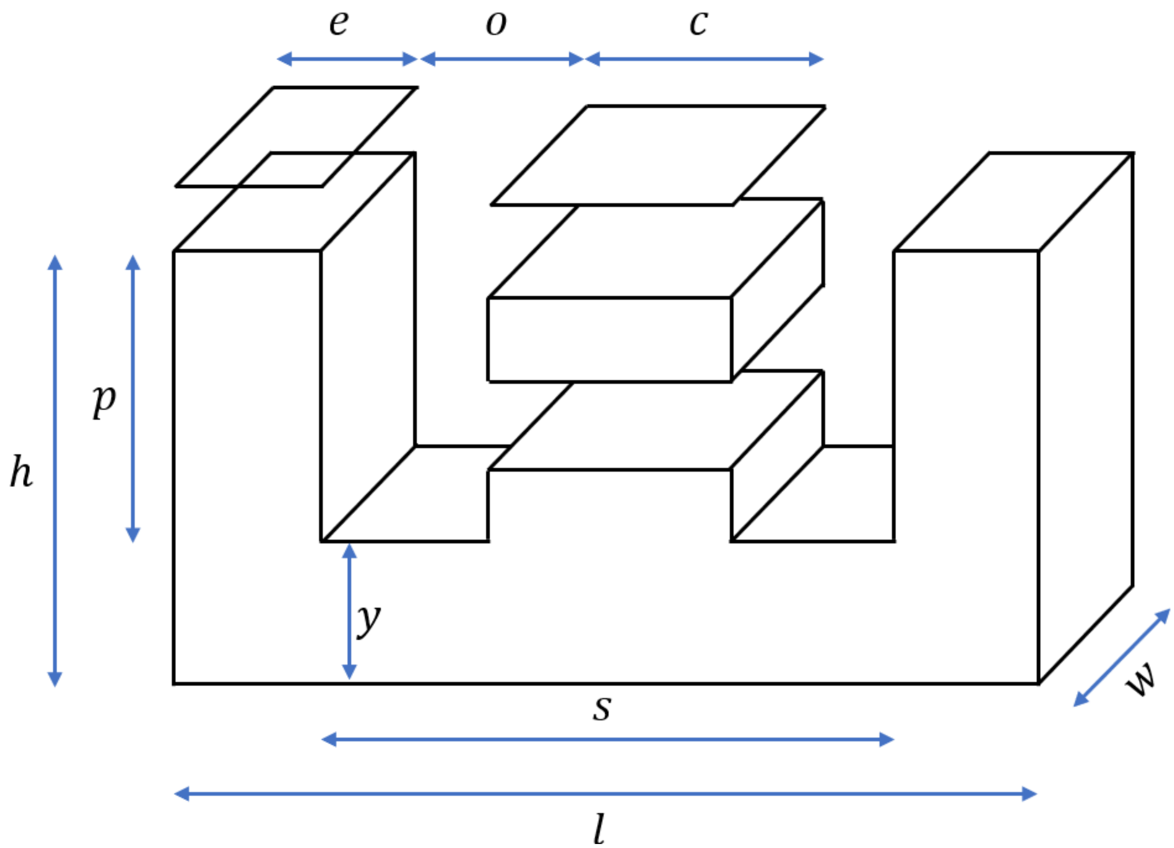
The described parameters of an E core half are depicted below:



If the "**Generate E cores from Us**" option is enabled in Configuration, two U cores can be stacked in parallel to generate an E core, as described in its corresponding section.

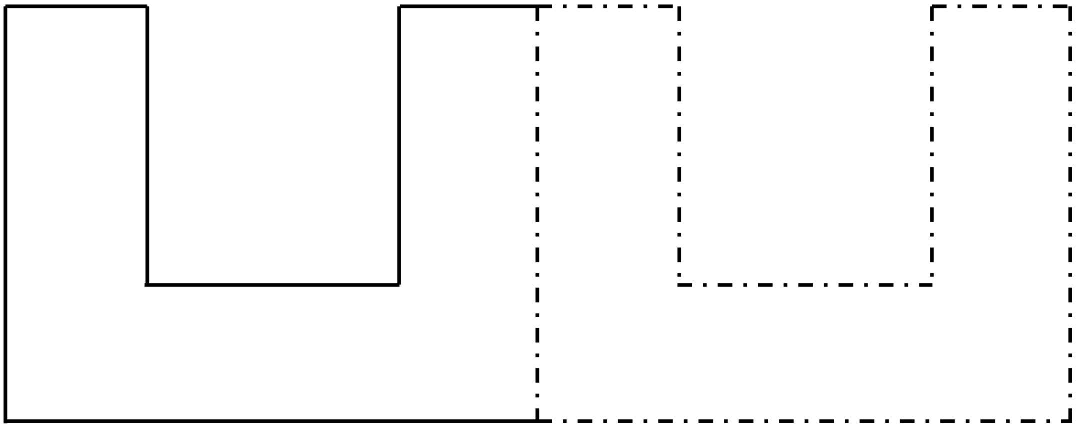
4.1.1.1 U Cores

In the current version, only E core geometries are allowed. The described parameters of an E core half are depicted below:

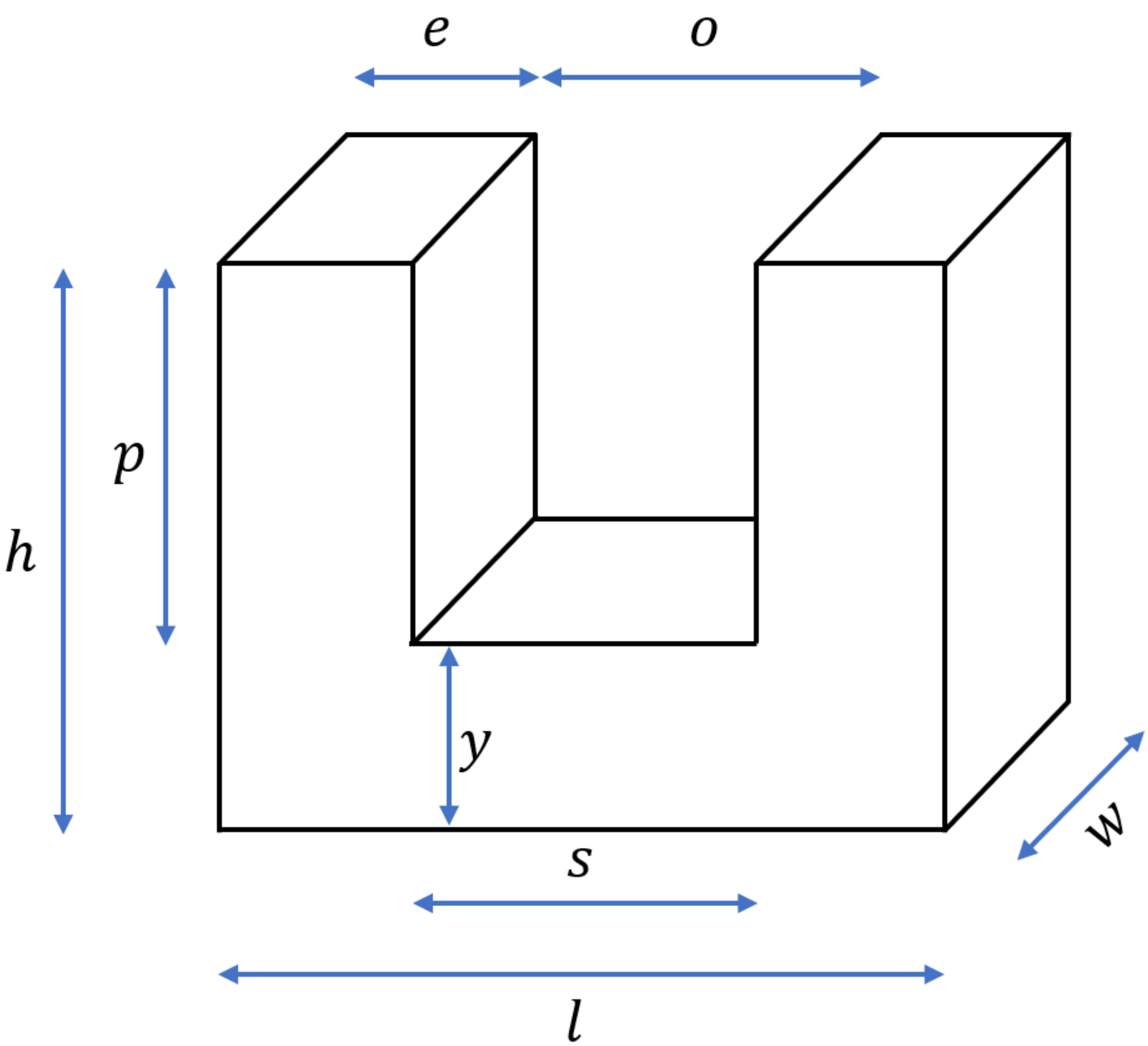


By default, only purely E cores are available when loading the core geometry database. However, to allow different (usually bigger) sizes, two U cores can be used in parallel to generate a bigger E core.

That option can be enabled with the "Generate E cores from Us" check box in the Configuration dialog. By doing so, the U cores present in the database will be loaded, and equivalent E cores will be generated from two of those cores in parallel, as depicted in the next figure:



In the database, the parameters that describe an U core are depicted below:



SmartNetics automatically handles the conversion from Us to Es. New entries will appear in the database as 2xU/xx/xx/xx and the needed calculations will be carried out: new equivalent core length, cross section, etc.

4.1.2 Core materials

The core materials database is composed of the following parameters:

- **Contemplated?**: choose if this particular material is to be considered for the design.
- **Material**: name of the material.
- **B sat (T)**: saturation flux density, in Tesla.
- **alpha (-)**: alpha exponent for Steinmetz Equation. See "Core loss models".
- **beta (-)**: beta exponent for Steinmetz Equation. See "Core loss models".
- **Kc (W/(HzTm^3))**: K coefficient for Steinmetz Equation. See "Core loss models".
- **Density (kg/m3)**: density of the material, in kg/m^3.
- **Initial permeability mu_i 25° (-)**: initial permeability of the material at 25°.
- **High amplitude permeability mu_a (-)**: values for the estimation of permeability dependance on amplitude. It consists of 3 parameters:
  - **mu_a**: peak value of mu_a.
  - **B**: flux density at which the peak value of mu_a is reached, in Tesla.
  - **T**: temperature for the provided data, in degrees Celsius.
- **Characteristic B-H points (T and A/m)**: characteristic points of the B-H curve for its estimation (These parameters do not affect the design, but are used when exporting to third party tools). It consists of 3 parameters:
  - **Hc** (coercivity): positive value of H when B is 0, in A/m.
  - **Br** (remanence): positive value of B when H is 0, in Tesla.
  - **Bs** (saturation): maximum value of B before reaching vacuum permeability, in Tesla.
- **Conductivity (S/m)**: electrical conductivity of the material, in S/m.
- **Thermal conductivity (W/(m·K))**: thermal conductivity of the material, in W/(m·K).
- **Specific heat (J/(kg·K))**: specific heat of the material, in J/(kg·K).
- **Thermal expansion coefficient (1/K)**: thermal expansion coefficient of the material, in 1/K. *Only used for the creation of the material in Ansys Icepak for thermal simulation.
- **Composition**: material type. There are two options:
  - **Ferrite**: This includes any material which inductance and losses are calculated using permeability and Steinmetz coefficients respectively. Ferrite, amorphous and noncrystalline materials fall in this definition.
  - **Powder**: This includes any material which inductance and losses are calculated using A_L and Magnetics' parameters respectively. Iron powder materials fall in this definition.
- **H_DC mu compensation (H in Oe)**: Only for Magnetics' Iron powder. Parameters for the compensation of permeability due to DC field. a, b and c are provided by Magnetics.

- **BH curve coefficients (H in Oe)**: Only for Magnetics' Iron powder. Parameters for precise calculation of B and H. a, b, c, d, e and x are provided by Magnetics.
  - This approximation is only used if "Precise B period calculation (Magnetics)" is active in the Models tab.

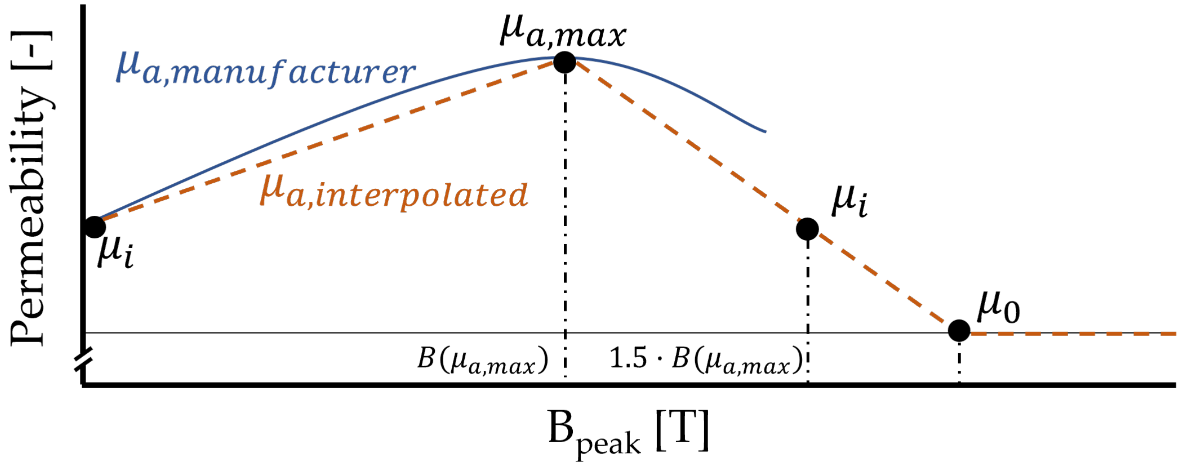
4.1.2.1 High amplitude permeability

In a real material, the permeability depends (amongst other things) on the point of the BH curve at which the material is operating. This means that there is not a single inductance value for the device during a full current or voltage period.
However, when designing a magnetic component, usually a single inductance value is used as a design objective, since describing a device by an infinite number of inductance values is impractical.

To overcome this issue, a single value, called *High amplitude permeability*, usually referred to as μ_a, can be used. This parameter allows the definition of a single-value inductance for the device during the whole period. It is defined to ensure that, for a given period, the real values of peak B and H fields (minimum and maximum) are reached, which in turn ensures that the real peak values of voltage and current (minimum and maximum) are reached.

Thanks to this definition, the behavior of the device's voltage and current maximums within a period can be studied (including saturation), without the need to incorporate the changes in the waveforms as the permeability does.

Since the value for μ_a depends on B, the manufacturers usually give it as curves. To ease the process for the user, in SmartNetics only the maximum value for μ_a is needed, along with the B value at which it occurs. From only those two values, a curve is reproduced as shown below in a dashed line (the blue line is the real one, provided by the manufacturer):



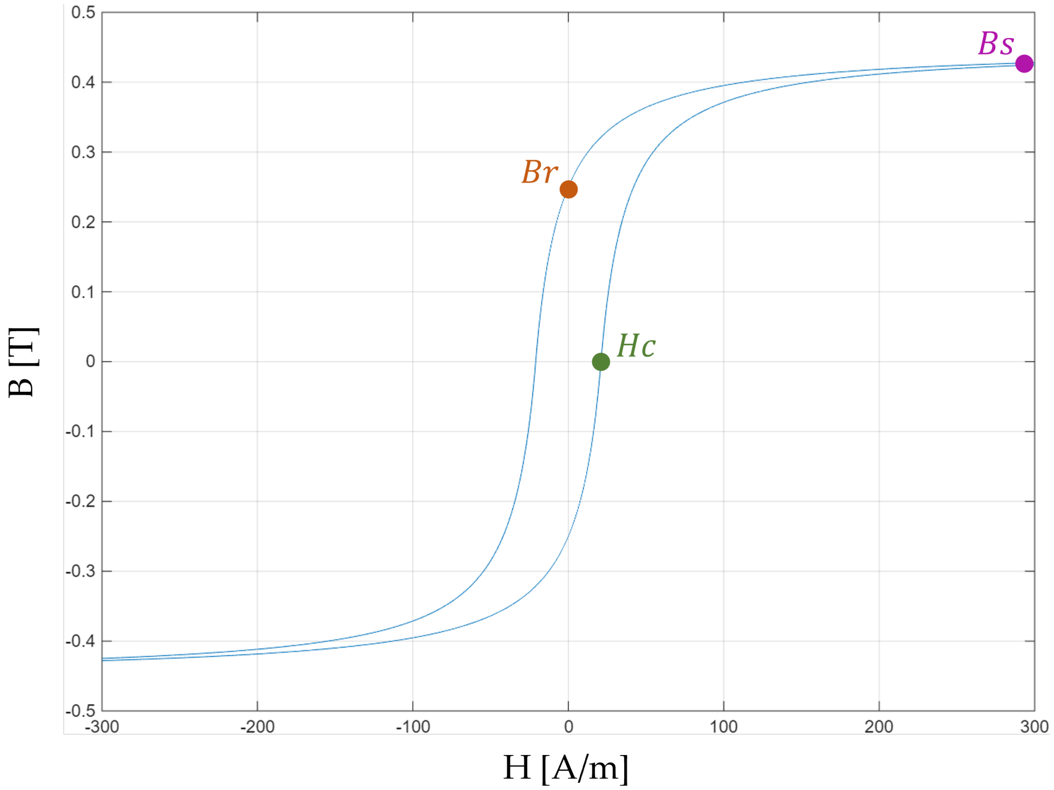
In SmartNetics, as shown in the figure, the curve is defined as:

- Linearly growing from μ_i at B=0 to μ_a at B=B($\mu_{a,max}$).
- Linearly decreasing from μ_a at B=B($\mu_{a,max}$) to μ_i at B=1.5B($\mu_{a,max}$).
- Linearly decreasing with a constant slope from μ_i at B=1.5B($\mu_{a,max}$) to μ_0 at the value of B at which vacuum permeability is reached.
- μ_0 for any B field higher than that

4.1.2.2 Characteristic B-H points

There are 3 main parameters for a material BH curve:

- **Hc** (coercivity): positive value of H when B is 0, in A/m.
- **Br** (remanence): positive value of B when H is 0, in Tesla.
- **Bs** (saturation): maximum value of B before reaching vacuum permeability, in Tesla.



The BH curve of some materials is provided by the manufacturers, but usually as a static figure in a PDF file. To ease the description for the user, only the main 3 parameters are needed in SmartNetics, from which an approximate BH curve is generated.

From these 3 points, the full BH curve is described by dividing it in two different branches:

- A lower branch, which is the path taken when the H values are increasing.
- An upper branch, which is the path taken when the H values are decreasing.

An approximation for the B field in each of those branches is provided in the following reference: Chan, J. H., Vladimirescu, A., Gao, X. C., Liebmann, P., & Valainis, J. (1991). Nonlinear transformer model for circuit simulation. IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems, 10(4), 476-482.

From the 3 values shown above, the required branches are approximated by:

$$B_{lower} = B_s \frac{H - H_c}{|H - H_c| + H_c\left(\frac{B_s}{B_r} - 1\right)} + \mu_0 H$$

$$B_{upper} = B_s \frac{H + H_c}{|H + H_c| + H_c\left(\frac{B_s}{B_r} - 1\right)} + \mu_0 H$$

*These parameters do not affect the design, but are used when exporting to third party tools.

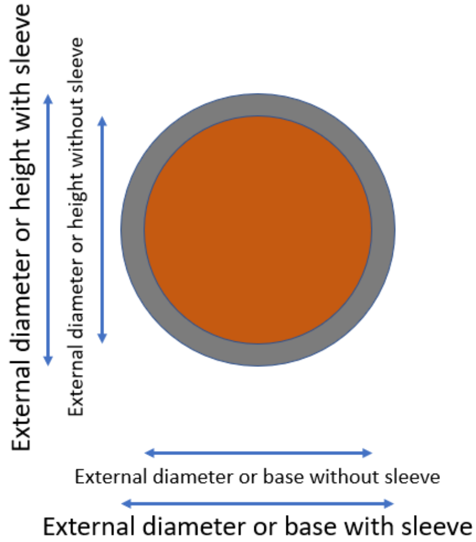
4.1.3 Conductors

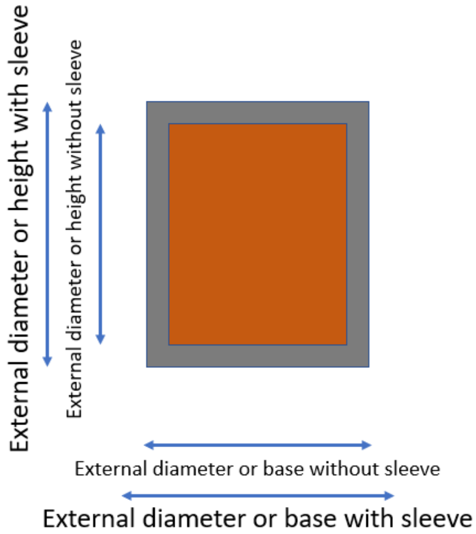
The conductors database is composed of the following parameters:

- **Contemplated?**: choose if this particular geometry is to be considered for the design.
- **Name**: name for the conductor.
- **Type**: type of conductor, Solid or Litz.
- **Conductor geometry**: geometry of every conductor: Round or Rectangular. Notice that, for Litz wire, conductors can be round while the wire as a whole is rectangular. See image below.
- **External geometry**: geometry of the wire: Round or Rectangular. Notice that, for Litz wire, conductors can be round while the wire as a whole is rectangular. See image below.
- **Conductors**: number of conductors that conform the wire. 1 for Solid wire or more for Litz wire.
- **Single diameter or base (m)**: diameter (for round conductors) or base (for rectangular conductors) of a single strand, in meters. For Solid wire conductors this is the same as "External diameter or base without sleeve (m)".
- **Single diameter or height (m)**: diameter (for round conductors) or height (for rectangular conductors) of a single strand, in meters. For Solid wire conductors this is the same as "External diameter or height without sleeve (m)".
- **External diameter or base with sleeve (m)**: diameter (for round conductors) or base (for rectangular conductors) of the whole wire, including its sleeve, in meters. See image below.
- **External diameter or height with sleeve (m)**: diameter (for round conductors) or height (for rectangular conductors) of the whole wire, including its sleeve, in meters. See image below.
- **External diameter or base without sleeve (m)**: diameter (for round conductors) or base (for rectangular conductors) of the whole wire, without its sleeve, in meters. See image below.
- **External diameter or height without sleeve (m)**: diameter (for round conductors) or height (for rectangular conductors) of the whole wire, without its sleeve, in meters. See image below.
- **Density (kg/m)**: density of the wire, in kg/m.
- **Material**: conducting material.
- **Sleeve material**: sleeve material.
- **Price (€/kg)**: price of the wire, in €/kg.

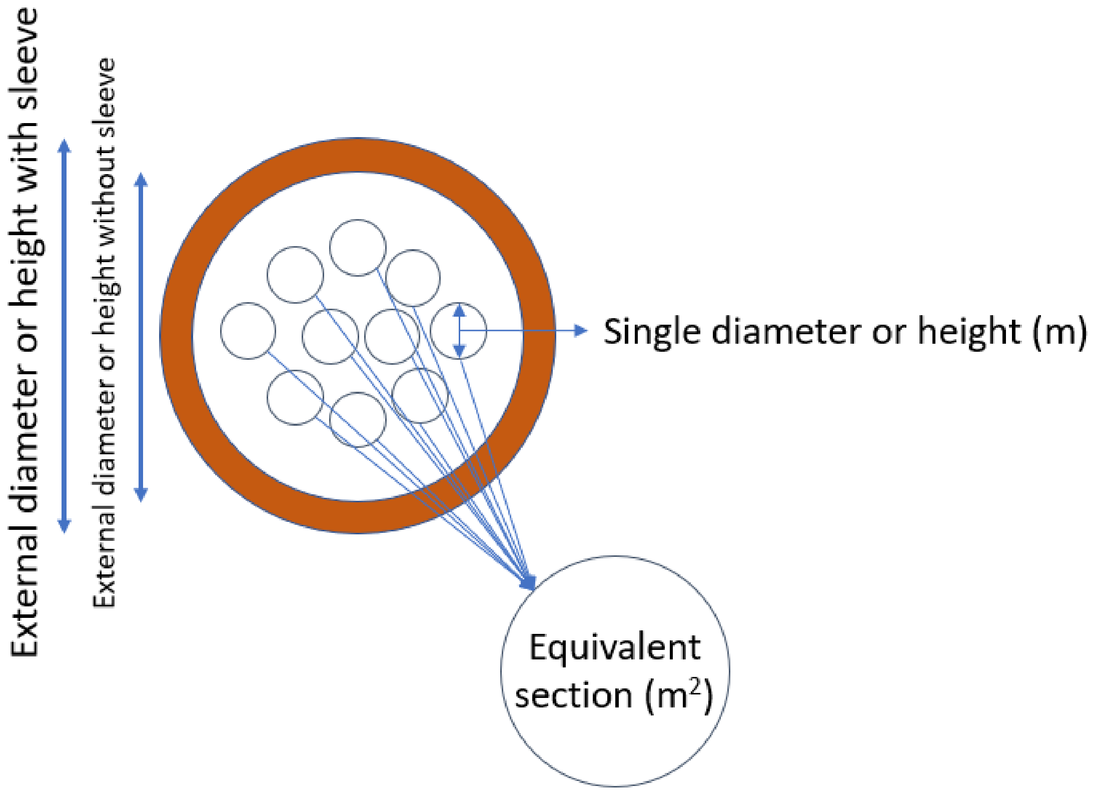
- **Equivalent section (m2)**: equivalent conducting section of the wire, in m^2. This parameter is not provided in the database but is calculated upon loading.

Main geometric parameters of rigid wire:





Main geometric parameters of Litz wire:



4.1.4 Conductor materials

The conductor materials database is composed of the following parameters:

- **Contemplated?**: choose if this particular material is to be considered for the design.
- **Material**: name for the conducting material.
- **Conductivity @ 20° (S/m)**: conductivity of the material at 20 °C.
- **Temperature coefficient for resistivity (1/K)**: coefficient by which the resistivity increases for every degree over 20 °C (This parameter only takes effect if "Temperature feedback to loss calculation" is selected in the thermal model configuration, otherwise, the resistivity at 20° is used). Resistivity follows the next expression (being α_{20} the temperature coefficient):

$$\rho_T = \rho_{20} \cdot \left[1 + \alpha_{20} \cdot (T - 20^{\circ}C)\right]$$

- **Density (kg/m)**: density of the material, in kg/m.
- **Thermal conductivity (W/(m·K))**: thermal conductivity of the material, in W/(m·K).
- **Specific heat (J/(kg·K))**: specific heat of the material, in J/(kg·K).
- **Thermal expansion coefficient (1/K)**: thermal expansion coefficient of the material, in 1/K. *Only used for the creation of the material in Ansys Icepak for thermal simulation.

4.1.5 Insulators

The insulators database is composed of the following parameters:

- **Contemplated?**: choose if this particular material is to be considered for the design.
- **Material**: insulator material.
- **Name**: name for a specific insulator. Notice that two insulators can be of the same material but have different properties due to different thicknesses.
- **Dielectric strength (V/m)**: dielectric strength of the insulator, in V/m.
- **Dielectric strength for partial discharges (V/m)**: dielectric strength of the insulator to ensure no partial discharges, in V/m.
- **Dielectric constant, permittivity (-)**: dielectric constant (permittivity) of the material.
- **Dissipation factor, loss tangent**: loss tangent (dissipation factor) of the material.
- **Conductivity (S/m)**: conductivity of the material, in S/m.
- **Thickness (m)**: thickness of every sheet, in meters.
- **Density (kg/m3**: density of the material, in kg/m^3.
- **Thermal conductivity (W/(m·K))**: thermal conductivity of the material, in W/(m·K).
- **Specific heat (J/(kg·K))**: specific heat of the material, in J/(kg·K). *Only used for the creation of the material in Ansys Icepak for thermal simulation.

- **Thermal expansion coefficient (1/K)**: thermal expansion coefficient, in 1/K. *Only used for the creation of the material in Ansys Icepak for thermal simulation.

4.1.6 Sleeves

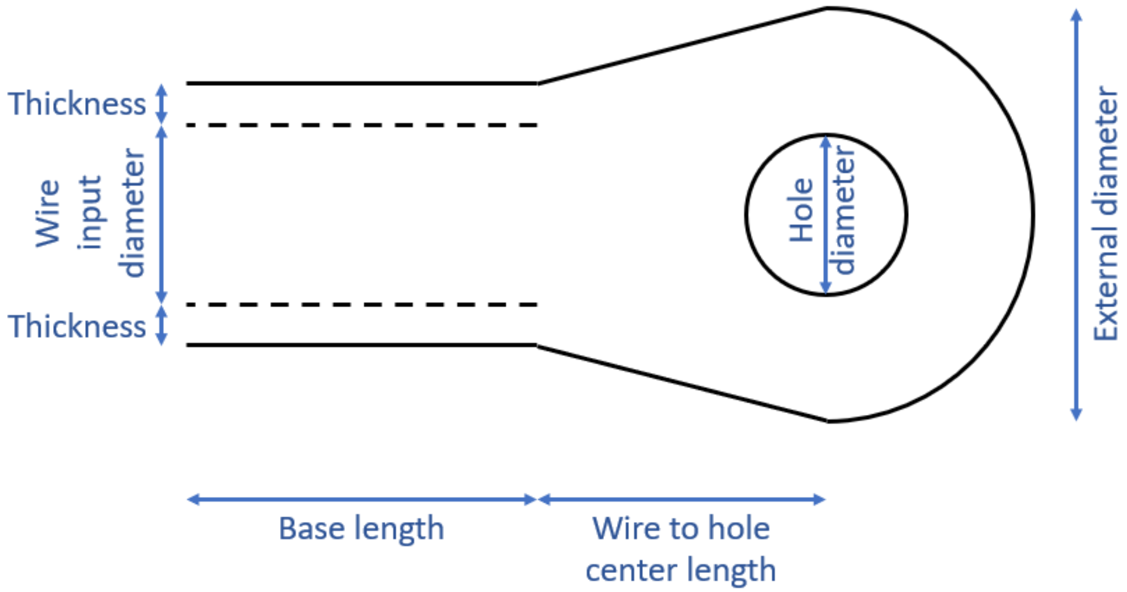
The sleeves database is composed of the following parameters:

- **Contemplated?**: choose if this particular material is to be considered for the design.
- **Material**: sleeve material.
- **Density (kg/m3)**: density of the material, in kg/m^3.
- **Thermal conductivity (W/(m·K))**: thermal conductivity of the material, in W/(m·K).
- **Specific heat (J/(kg·K))**: specific heat of the material, in J/(kg·K). *Only used for the creation of the material in Ansys Icepak for thermal simulation.
- **Thermal expansion coefficient (1/K)**: thermal expansion coefficient, in 1/K. *Only used for the creation of the material in Ansys Icepak for thermal simulation.

4.1.7 Connectors

The connectors database is composed of the following parameters:

- **Contemplated?**: choose if this particular material is to be considered for the design.
- **Model**: model of the connector.
- **Manufacturer reference**: manufacturer reference for that particular model.
- **Hole metric**: metric of the hole. This parameter is only used as reference, "Hole diameter" is the one used for calculations.
- **Hole diameter**: diameter of the hole. See image below.
- **Wire input diameter (m)**: diameter of the wire input, in meters. See image below.
- **External diameter (m)**: external diameter of the terminal, in meters. See image below.
- **Wire to hole center length (m)**: distance from the end of the wire to the center of the hole, in meters. See image below.
- **Base length (m)**: distance from the beginning to the end of the wire, in meters. See image below.
- **Thickness (m)**: thickness of the connector, in meters. See image below.
- **Price (€)**: price of a single connector, in €.
- **Manufacturer**: manufacturer of the connector.



*In the current version Connectors are not used in the design

4.1.8 Editing

In the current version, databases cannot be edited inside SmartNetics.
Databases can still be edited to modify an entry or to add new ones, in two ways:

- Editing the .csv (Comma Separated Values) files
- Using specific Excel files

Here, only a brief explanation is provided, but the user is highly encouraged to use the tutorial provided at [SmartNetics' website] for the process of editing the databases.

- **Editing the .csv (Comma Separated Values) files:**
  The databases inside SmartNetics are loaded from .csv files, that are accessible for the user, and that can be modified to change any value of a given entry or to add new entries.
  There is a separate .csv file for every database and their contents follow the same structure:
  1. A first row of column names
  2. A row of values separated by commas for every entry

  The format of every value depends on the data type. They can be strings, boolean, floating point number or even structures. For example:

```
Contemplated?,Material,Conductivity @ 20° (S/m),Temperature coefficient for resistivity (1/K),Density (kg/m3),Thermal conductivity (W/(m·K)),Specific heat (J/(kg·K)),Thermal expansion
TRUE,Copper,58000000,0.00393,8933,400,385,1.77e-05
TRUE,Aluminium,35850000,0.004308,2700,237.5,951,2.33e-05
```

  To edit any of them, simply change the value in the corresponding .csv file.

Some data types are very complex, since they have multiple parameters inside a single structured value, so adding new entries can be difficult. To do so, it is highly advised to copy an existing row (as similar as the one to be added as possible), and paste it at the end of the document. After doing so, the user can change the name of the newly added component and modify any existing value.

*Please, be careful, since two parameters cannot have the same name in the database.

- **Using specific Excel files:**

To help the user when modifying entries or adding new ones, an alternative way is provided.

In SmartNetics' website the user can find the same version of the databases, but in a more friendly Excel file. Where every entry, even the ones that are structures, can be displayed. For example:

☐	Kool Mu 14u	1	1.541	1.988	0.698019605	a = 0.01 b = 3.066e-7 c = 1.85	a = 5.216e-2 b = 1.507e-2 c = 4.323e-4 d = 1.036e-1 e = 5.174e-4 x = 1.952	5800	14
☐	Kool Mu 26u	1	1.541	1.988	0.767583333	a = 0.01 b = 4.581e-7 c = 1.868	a = 2.710e-2 b = 9.151e-3 c = 4.036e-4 d = 7.636e-2 e = 3.986e-4 x = 1.515	5800	26
☐	Kool Mu 40u	1	1.541	1.988	0.815467954	a = 0.01 b = 7.684e-7 c = 1.904	a = 4.990e-2 b = 1.537e-2 c = 5.792e-4 d = 7.263e-2 e = 5.542e-4 x = 1.689	5800	40

To modify any entry, simply edit the corresponding field. To add a new one, it is highly advised to copy an existing row (as similar as the one to be added as possible), and paste it a t the end of the document. After doing so, the user can change the name of the newly added component and modify any existing value.

*Please, be careful, since two parameters cannot have the same name in the database.

As stated before, SmartNetics loads the databases from .csv files. If the Excel approach is followed, the databases have to be translated from .xlsx files to their proper .csv counterparts. To allow this transition, the user has a small tool available in our web. This tool is described in its corresponding section.

4.1.8.1 Excel to csv

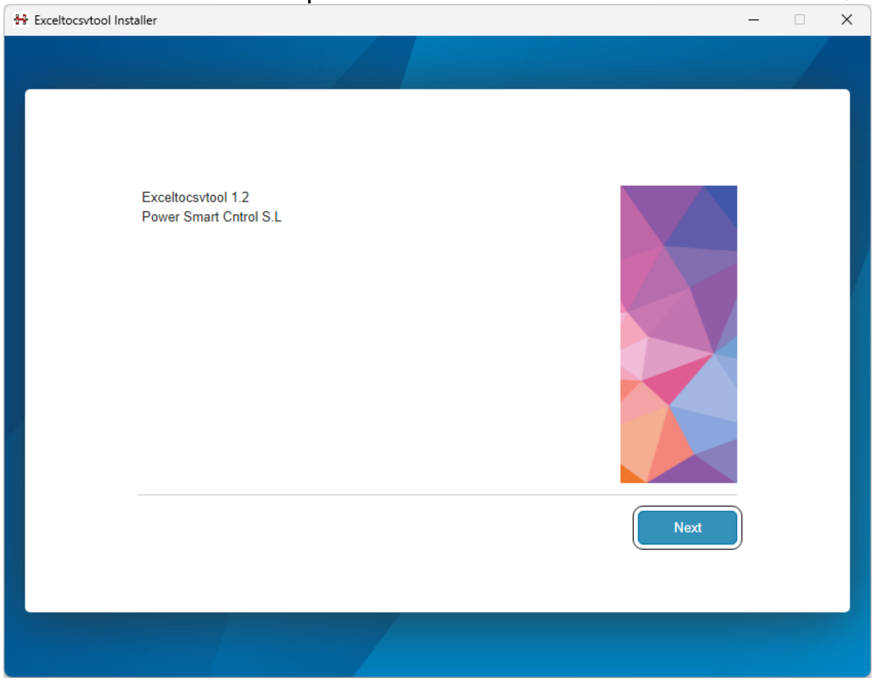
SmartNetics loads the databases from .csv files that can be edited by the user.

Since editing .csv files can be difficult, we provide a tool to generate them from easy-to-read Excel files.

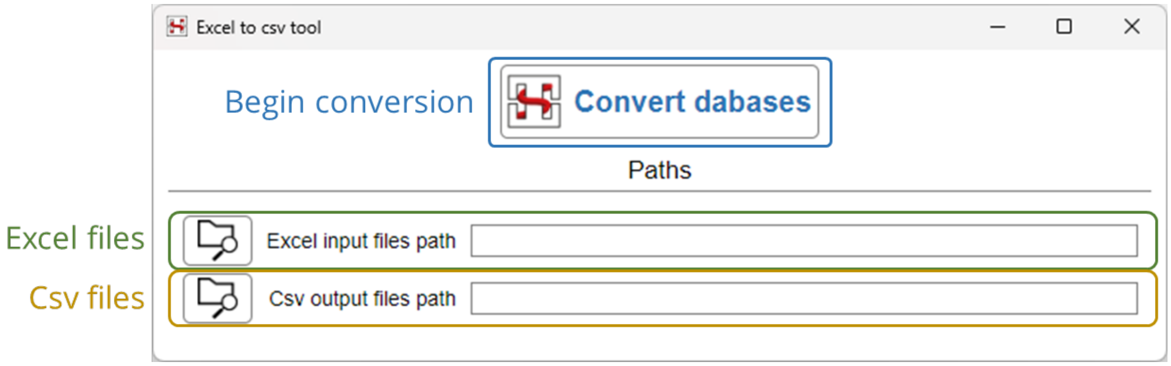
The user can download those Excel files from SmartNetics' website and edit them, as stated in the corresponding section. Once they are edited, we provide a free tool to convert them from .xlsx to the .csv files SmartNetics can use.

That tool is available at SmartNetics' website and can be downloaded for free.

Once downloaded, the user can begin its installation.
This small standalone tool is developed in Matlab, but the user does not need a Matlab license to run it.
*Although the tool is relatively small, the first time it is installed it needs to install not Matlab itself but a compressed version of its needed functions, so it can take a while.



Once installed, the user can launch it. It consists of a single dialog divided in three parts:



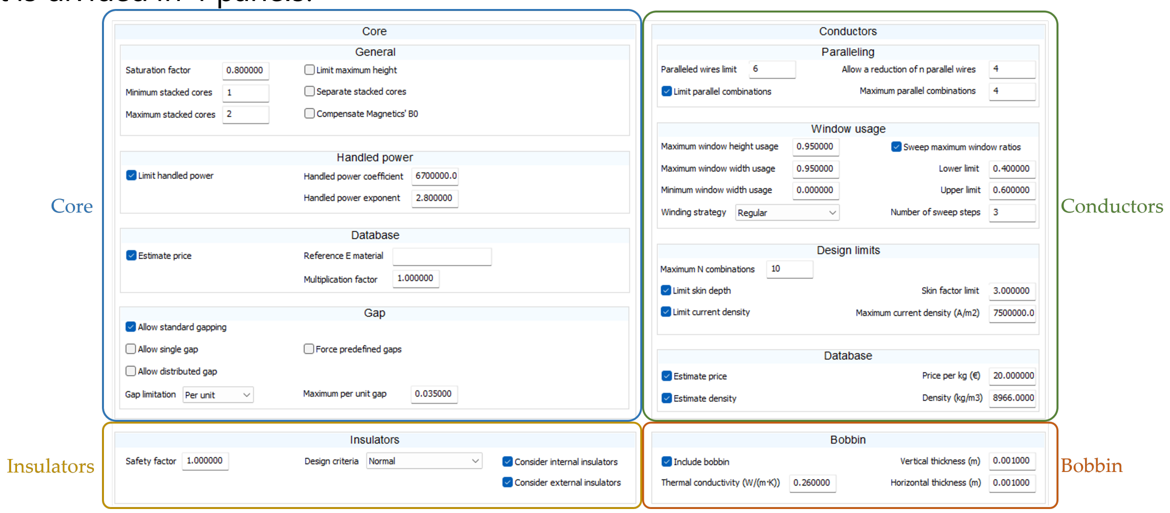
- **Begin conversion**: After selecting a path from where the Excel files are going to be loaded and a path for the csv files to be generated, the automatic conversion can begin.

- **Excel files**: Path where the Excel files are stored. Every database file is needed, whether they were modified or not:
  - Core geometries.xlsx
  - Core materials.xlsx
  - Conductors.xlsx
  - Conductor materials.xlsx
  - Insulators.xlsx
  - Sleeves.xlsx
  - Connectors.xlsx
- **Csv files**: Path where the .csv files are going to be generated. The user can select the folder where SmartNetics is installed so data bases are automatically loaded at launch, or a different one and then load them inside the application, as explained in its corresponding section.

4.2 Device parts

This tab allows the user to access every parameter related to the parts that compose the magnetic component.

It is divided in 4 panels:



Every field is described in its corresponding section:

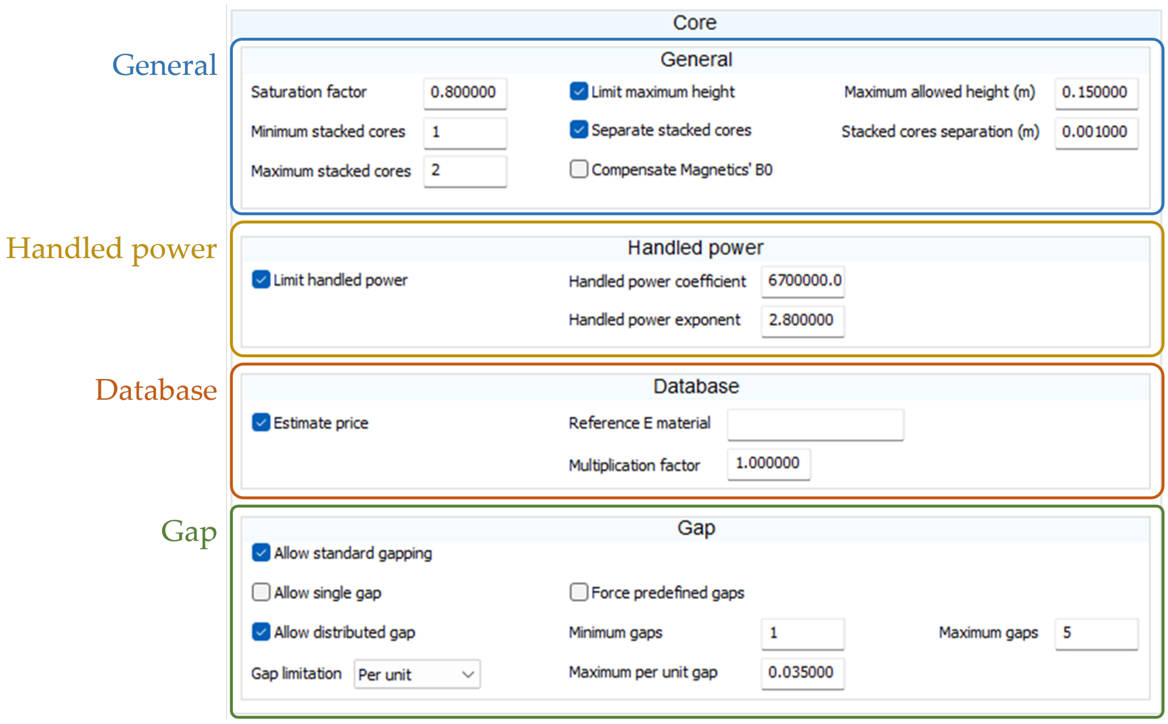
- Core
- Conductors
- Insulators
- Bobbin

4.2.1 Core

This panel groups the options related to the magnetic core.

The number of options present depend on the selected configuration and the device type (Inductor or transformer).

It is divided in four sections:



General

- **Saturation factor**: per-unit value of the saturation field that the component is allowed to reach.
- **Minimum stacked cores**: minimum number of cores to be stacked in parallel.
- **Maximum stacked cores**: maximum number of cores to be stacked in parallel.
- **Limit maximum height**: imposes a maximum height for the device. It enables the "Maximum allowed height (m)" field.
  - **Maximum allowed height (m)**: maximum height, in meters. Any taller device will not be considered valid.
- **Separate stacked cores**: imposes a distance between cores when they are stacked. It enables the "Stacked cores separation (m)".
  - **Stacked cores separation (m)**: horizontal distance between stacked cores, in meters.
- **Compensate Magnetics' B_0**: forces a H=0, B=0 crossing of the B-H curve for Magnetics' materials.
  This modifies the equation provided by Magnetics for the estimation of DC flux impact and allows a higher precision for low DC values in exchange for an accuracy reduction at high DC values.

Handled power

- **Limit handled power**: allows the speed-up of the design procedure by filtering out designs that are estimated to be unable to handle the required power. It enables the "Handled power coefficient" and "Handled power exponent" fields. Refer to the related topic for additional information.

Database

- **Estimate price**: estimate the price of the cores that do not have a particular value in the database.
- **Reference E material**: material to be used as reference for the price calculation. If empty, the material with the most entries is used.
- **Multiplication factor**: factor by which the estimated price is escalated.

Gap (only for inductors, gapped transformers are not supported in the current version). See the Gap section for a graphical representation of the provided options.

- **Allow standard gapping**: allows the creation of inductors with the same gap on every leg.
- **Allow single gap**: allows the creation of inductors with a single gap in the central leg.
- **Force predefined gaps**: forces the use of gaps already defined in the database. It only affects single and distributed gaps, but not standard ones.
- **Allow distributed gap**: allows distributed gaps in the central leg. Only odd numbers are used, to ensure top and bottom half-cores remain symmetric. It enables the "Minimum gaps" and "Maximum gaps" fields.
  - **Minimum gaps**: minimum number of gaps allowed for the design with distributed gaps.
  - **Maximum gaps**: maximum number of gaps allowed for the design with distributed gaps.
- **Gap limitation:** sets the restriction for the maximum gap length (distance between top and bottom half-cores). Options are:
  - **Per unit**: the maximum gap is a per-unit portion of the total core height (top+bottom half core heights). This allows the use of different maximum gaps for different core geometries.
  - **Absolute**: the maximum gap is defined by a length value, in meters. This imposes the same restriction independently of the core geometry.

In SmartNetics, the value used for the calculation is not the distance between cores, but the total length of air the flux lines would travel in straight lines.
This distance is called *l_g (m)* and is the one provided in the variable list. It relates to gap (distance between cores) differently depending on the gapping strategy:

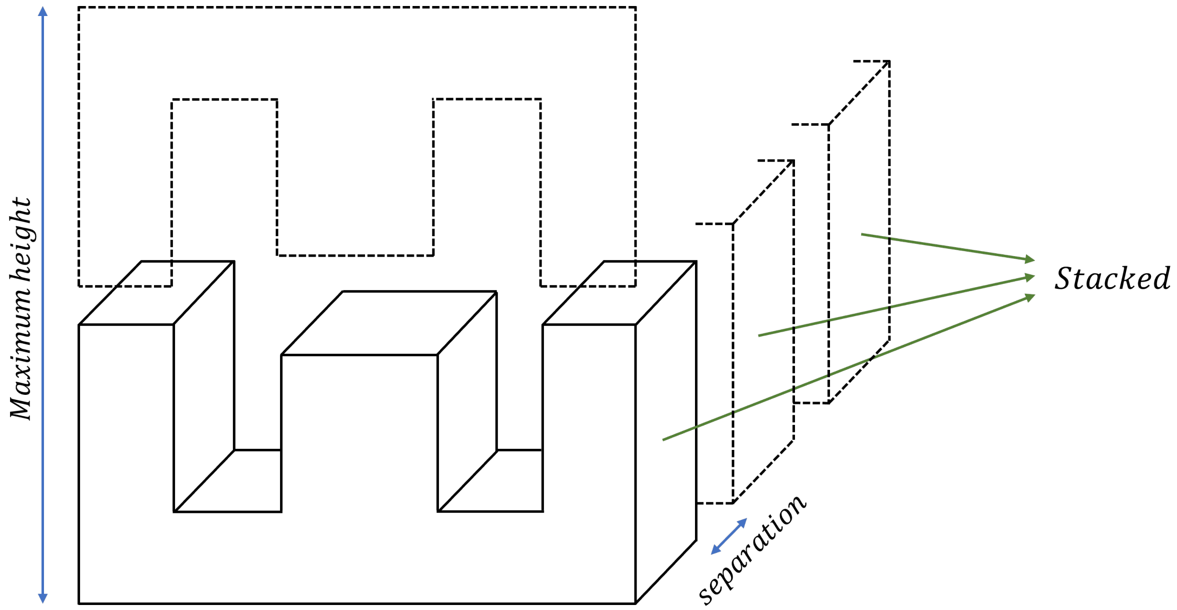
- **Standard gapping**: same gap for every leg. The total length of the magnetic path through the air (*l_g (m)*) is twice the distance between cores.

- **Single gap**: a single gap in the central columns. The total length of the magnetic path through the air (*l_g (m)*) is the distance between cores.
- **Distributed gaps**: a number of gaps in the central columns. The total length of the magnetic path through the air (*l_g (m)*) is the sum of every gap.

4.2.1.1 Stacking

Any number of cores may be stacked in parallel. The considered values are limited by the "Minimum stacked cores" and "Maximum stacked cores" fields.

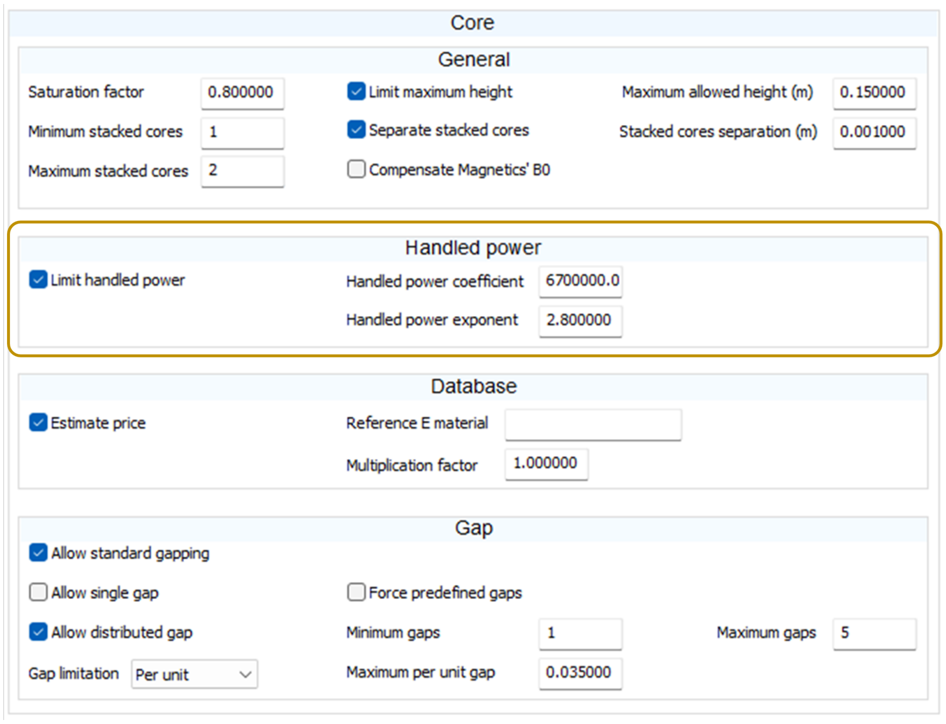
If "Separate stacked cores" is checked, a distance is added between each stacked core, as shown in the image below.



4.2.1.2 Limit handled power

This topic relates to the "Limit handled power" field in the Core section of "Device parts"

Handled power



Enabling this field speeds up the process by filtering out designs that are estimated to be unable to handle the required power. It enables the "Handled power coefficient" and "Handled power exponent" fields.

This coefficient and exponent are the A and gamma (γ) parameters of the next equation, presented in [Bossche2005]

$$S_{max,core} = A \cdot d_{max}^{\gamma}$$

The suggested default values of A=6.7e6 and gamma = 2.8 have been found to speed up the process without filtering out designs that would be valid.

[Bossche2005]: Valchev, V. C., & Van den Bossche, A. (2018). *Inductors and transformers for power electronics*. CRC press.

4.2.1.3 Gap

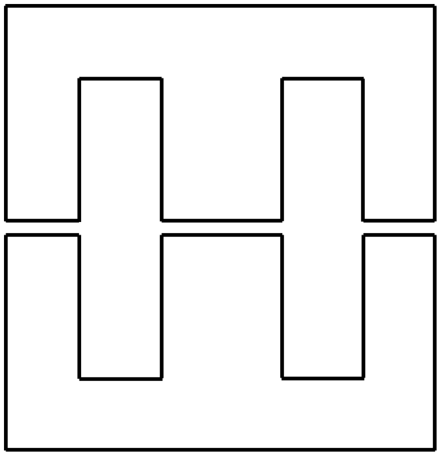
There are three gaping options (plus no gap).

In SmartNetics, the value used for the calculation is not the distance between cores, but the total length of air the flux lines would travel in straight lines.

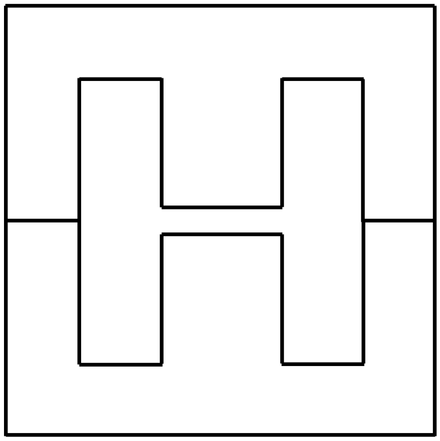
This distance is called *l_g (m)* and is the one provided in the variable list. It relates to gap (distance between cores) differently depending on the gapping strategy:

- **Standard gapping**: same gap for every leg. The total length of the magnetic path through the air (*l_g (m)*) is twice the distance between cores.

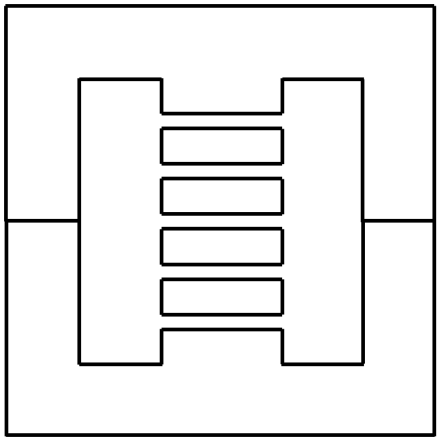
- **Single gap**: a single gap in the central columns. The total length of the magnetic path through the air (*l_g (m)*) is the distance between cores.
- **Distributed gaps**: a number of gaps in the central columns. The total length of the magnetic path through the air (*l_g (m)*) is the sum of every gap.



Standard gapping



Single gap



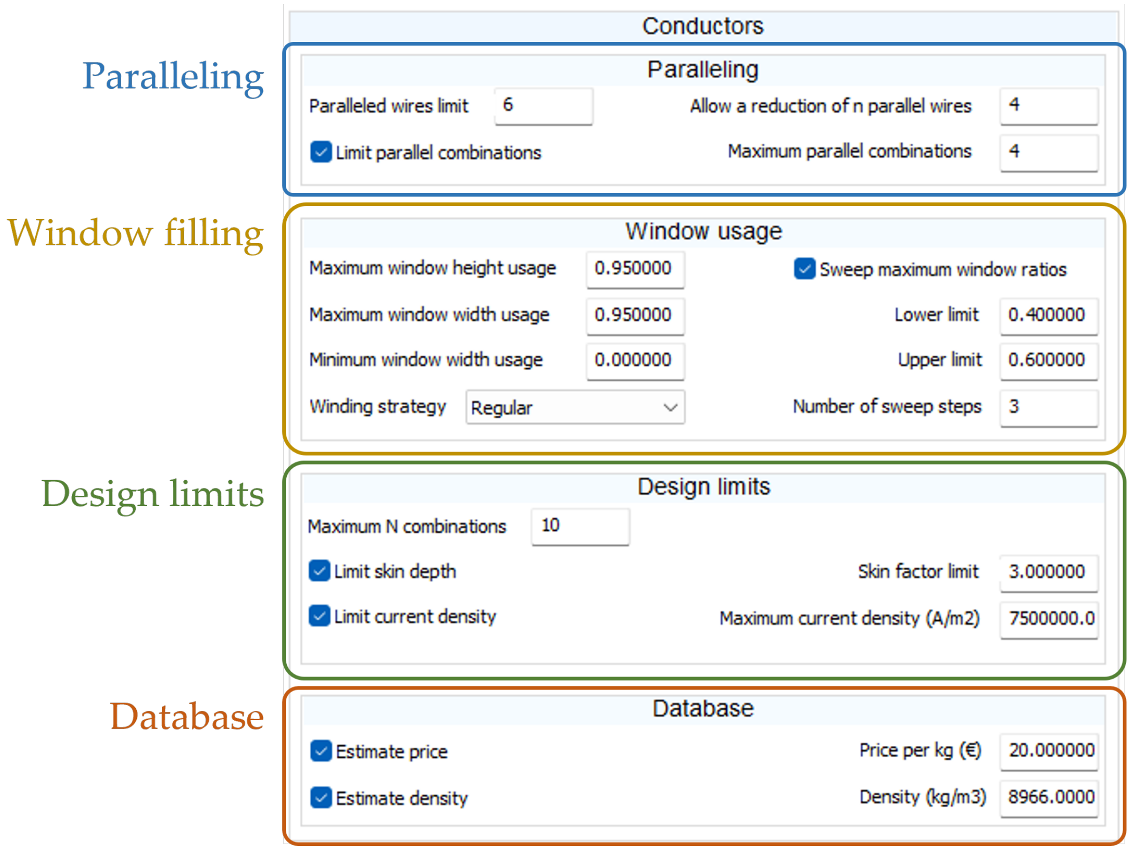
Distributed gaps

4.2.2 Conductors

This panel groups the options related to the conductors that make up the windings.

The number of options present depends on the selected configuration and the device type (Inductor or transformer)

It is divided in four sections:



Paralleling

Please, refer to the paralleling section for a detailed explanation of the paralleling strategy.

- **Paralleled wires limit**: establishes an absolute maximum limit number of wires in parallel.
  - There are more restrictions to the wires in parallel. The actual limit for a particular design also depends on the available window, the number of turns, etc.
- **Allow a reduction of n parallel wires**: allows the use of a number of parallel wires lower than the maximum.
  - A value of 0 means that only the highest number of wires that fills the available window is used.

- A higher value allows the reduction of that number by n. For example, if 10 wires in parallel would result in a total window filling, a value of 3 in this parameter allows designs with 10, 9 (10-1), 8 (10-2) and 7 (10-3).
  - A low number speeds up the design procedure by always using the maximum number of wires to reduce DC losses.
  - A higher number slows down the design procedure but allows designs with a lower number of wires in parallel, which can result in cheaper or easier-to-build designs.

- **Limit parallel combinations**: allows a reduction of the possible parallel combinations. By default, every possible rectangular combination of the paralleled wires is considered. For example, for six wires in parallel, the possible wire arrangements ([wires in horizontal, wires in vertical]) are 4:

  [1,6]
  [2,3]
  [3,2]
  [6,1]

if a limitation is established in **Maximum parallel combinations** to, for example, 3 of them, the remaining available arrangements would be:

  [1,6]
  [2,3]
  [6,1]

* The first and last ones have priority and then a linear distribution of the values in the middle.

Window filling

- **Maximum window height usage**: per-unit maximum filling of the window in the vertical direction.
  - A low value (<0.75) reduces the available space for copper and usually increases wire loss, but generates designs that are easier to build.
  - A high value (>0.9) increases the available space for copper and enables more designs, but some of them may be hard to manufacture.
- **Maximum window width usage**: per-unit maximum filling of the window in the horizontal direction.
  - A low value (<0.75) reduces the available space for copper and usually increases wire loss, but generates designs that are easier to build.
  - A high value (>0.9) increases the available space for copper and enables more designs, but some of them may be hard to manufacture.
- **Minimum window width usage**: per-unit minimum filling of the window in the horizontal direction.

- This field allows avoiding designs with a very low window usage, which usually result in devices that are bulkier and more expensive than needed. Usually, a value up to 0.5 gives good results.
- If this value is too high, some results may be discarded, and even a no-designs message may appear.

- **Sweep maximum window ratios allowed for primary** (only for transformers): allows a window sharing between primary and secondary windings different from 50/50. It enables the fields **Lower limit**, **Upper limit** and **Number of sweep steps**.
  - **Lower limit**: lower limit of the per-unit value of the window that is allowed, as maximum, for the primary winding.
  - **Upper limit**: upper limit of the per-unit value of the window that is allowed, as maximum, for the primary winding.
  - **Number of sweep steps**: number of possible values for the maximum per-unit value available for the primary winding.
  - For example, if the limits are established as:
    - Lower limit = 0.4
    - Upper limit= 0.6
    - Number of sweep steps= 3

      3 different values are used as a limit for the window available for the primary winding: 0.4, 0.5 and 0.6, which refer to 40%, 50% and 60% of the window.
  - A low value (<4) in this field allows a fast procedure by discarding designs that are similar to each other.
  - A high value (>6) slows down the procedure, but allows more designs, which could result in a better transformer.
- **Winding strategy** (only for transformers): transformer winding positioning. Two options are available:
  - Regular: the two windings go around the leg as tightly as possible. This strategy reduces wire length as much as possible and also reduces leakage inductances.
  - Max horizontal: the two windings are horizontally separated as much as the window allows. This strategy generates much greater leakage inductances and can be used to integrate inductor and transformer in a single device.

Design limits

- **Maximum N combinations**: maximum different values for the number of turns that are considered. This parameter does not limit the maximum number of turns, only the number of values considered. For example, if a particular window can fit up to 50 turns, but the maximum combinations are set to 5, only five numbers of turns are considered: 10, 20, 30, 40 and 50 turns.
  - A low value (<10) in this field allows a fast procedure by discarding designs that are similar to each other.

  - A high (>20) value slows down the procedure, but allows more designs, which could result in a better inductor or transformer.
- **Limit skin depth**: limits the available wires to only those that have a radius smaller than X times the skin depth.
  - The X value is set by the **Skin factor limit** field.
    - A low value (<5) speeds up the procedure by avoiding conductors with a low copper usage.
    - A higher value (>10) slows down the procedure, but provides designs that may be more convenient in other aspects, like cost, manufacturing or heat dissipation.
- **Limit current density**: limits the available wires to only those that have a section big enough to ensure the current density does not go over a certain value.
  - Said value is set by the **Maximum current density (A/m2)** field.
    - A low value (<5 A/m2) speeds up the procedure by discarding conductors that are expected to reach too high temperatures.
    - A high value (>10 A/m2) allows designs that are smaller or use less conductor material but slows down the procedure.

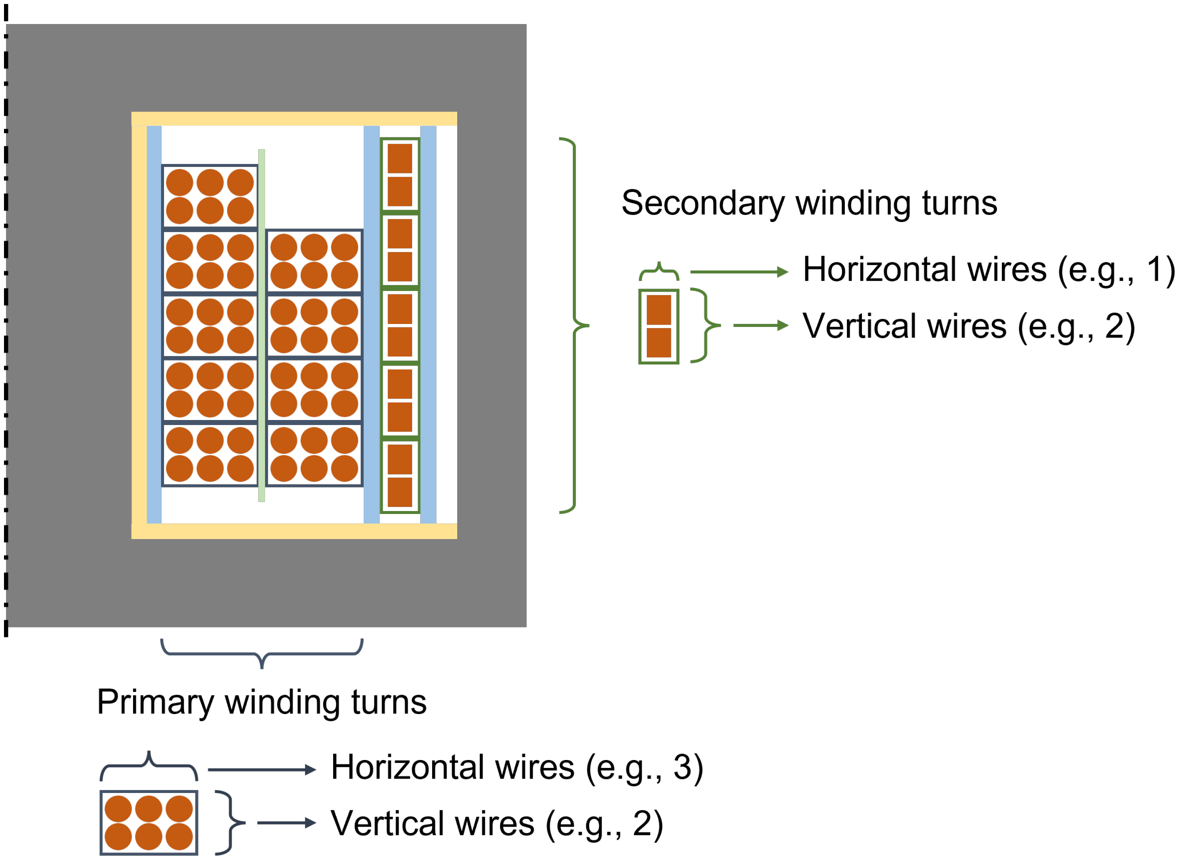
Database

- **Estimate price**: estimate the price of the conductors that do not have a particular value in the database.
- **Price per kg (€)**: price per kilogram of conductor to be used for the price estimation.
- **Estimate density**: estimate the density of the conductors that do not have a particular value in the database.
- **Density (kg/m3)**: density of the conductor to be used for the density estimation.

4.2.2.1 Paralleling

Every winding is composed of one or more wires in parallel.

SmartNetics uses a rectangular paralleling strategy for each winding. The position of the wires is fixed to fill a rectangular pattern, where every turn is composed by an m x n (horizontal x vertical) matrix of wires, as shown in the next figure:



The number of turns for a winding is provided in the variables N (for inductors), N1 and N2 (for transformers). Additionally, the maximum number of turns of the layers of a particular winding is provided in the variables BpL, BpL_1 and BpL_2, which refer to Bundles per Layer of winding.

Only the number of the fully occupied layers is provided. For example, for the design shown above, the primary winding Bundles per Layer (BpL_1) is 5, even though the outmost layer has only 4 turns.

Several numbers of wires in parallel can be considered for the same design, from a maximum to a minimum. The considered numbers of wires in parallel are limited:

- To a maximum imposed by the most restrictive criterion between “Parallel wires limit” and the physical limit imposed by the window.
- To a minimum imposed by the maximum minus the “Allow a reduction of n parallel wires” field.

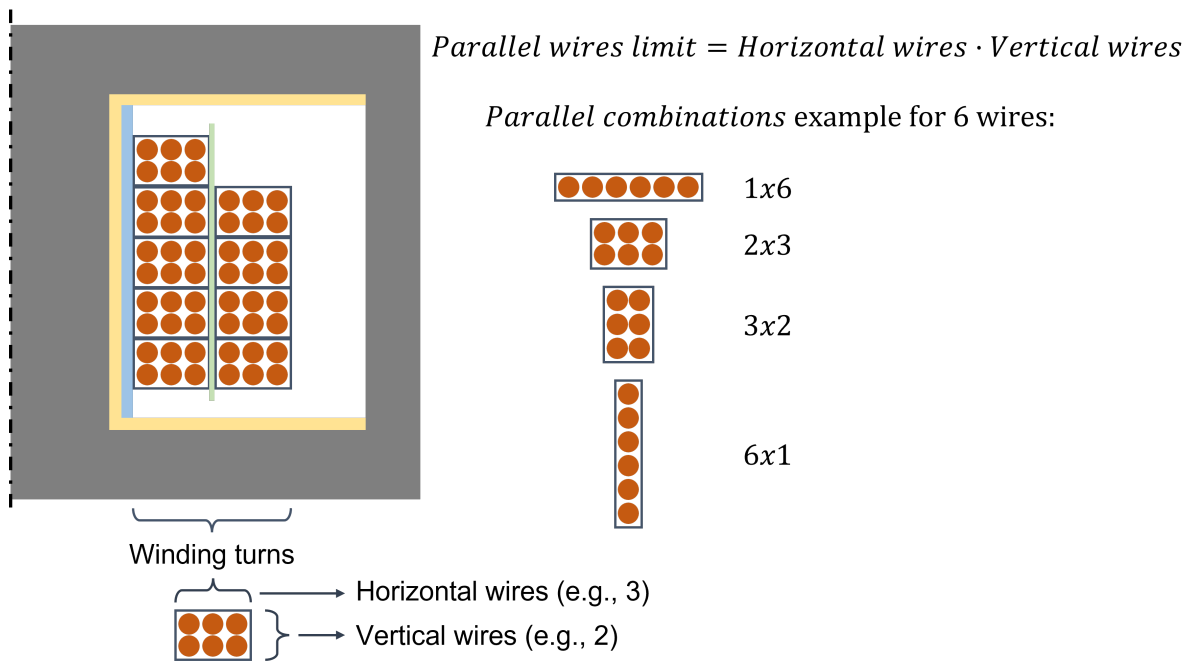
For example, if the maximum number of parallel wires that fit the window is 8, but the value set for "Paralleled wires limit" is 6, 6 will be taken as the maximum for the design.

The minimum number of paralleled wires would then be 6 minus the value in "Allow a reduction of n parallel wires", for example, if that parameter is set to 2, designs with 6, 5 and 4 wires in parallel will be considered.

Take into account that the limit imposed by the available window depends on the geometry of core and wire, so for some core-wire combinations the limit will be imposed by the window and for other by the "Paralleled wires limit" field. The minimum number is recalculated every time for every combination.

For a given number of wires in parallel, different stacking strategies can be considered, depending on the number of wires in horizontal and vertical that compose every turn. An example is shown in the next figure.

If the "Limit parallel combinations" options is checked, only a set of the possible combinations is considered. This speeds up the design process, but valid solutions may be left out.



The field "*Limit parallel combinations*" allows a reduction of the possible parallel combinations. By default, every possible rectangular combination of the paralleled wires is considered. For example, for six wires in parallel, the possible wire arrangements ([wires in horizontal, wires in vertical]) are 4:

[1,6]
[2,3]

[3,2]
[6,1]

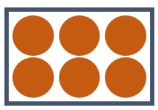
if a limitation is established in **Maximum parallel combinations** to, for example, 3 of them, the remaining available arrangements would be:

[1,6]
[2,3]
[6,1]

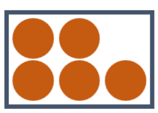
* The first and last ones have priority and then a linear distribution of the values in the middle.

Notice: In the current implementation, only fully-occupied rectangles are used:

Valid



Not valid



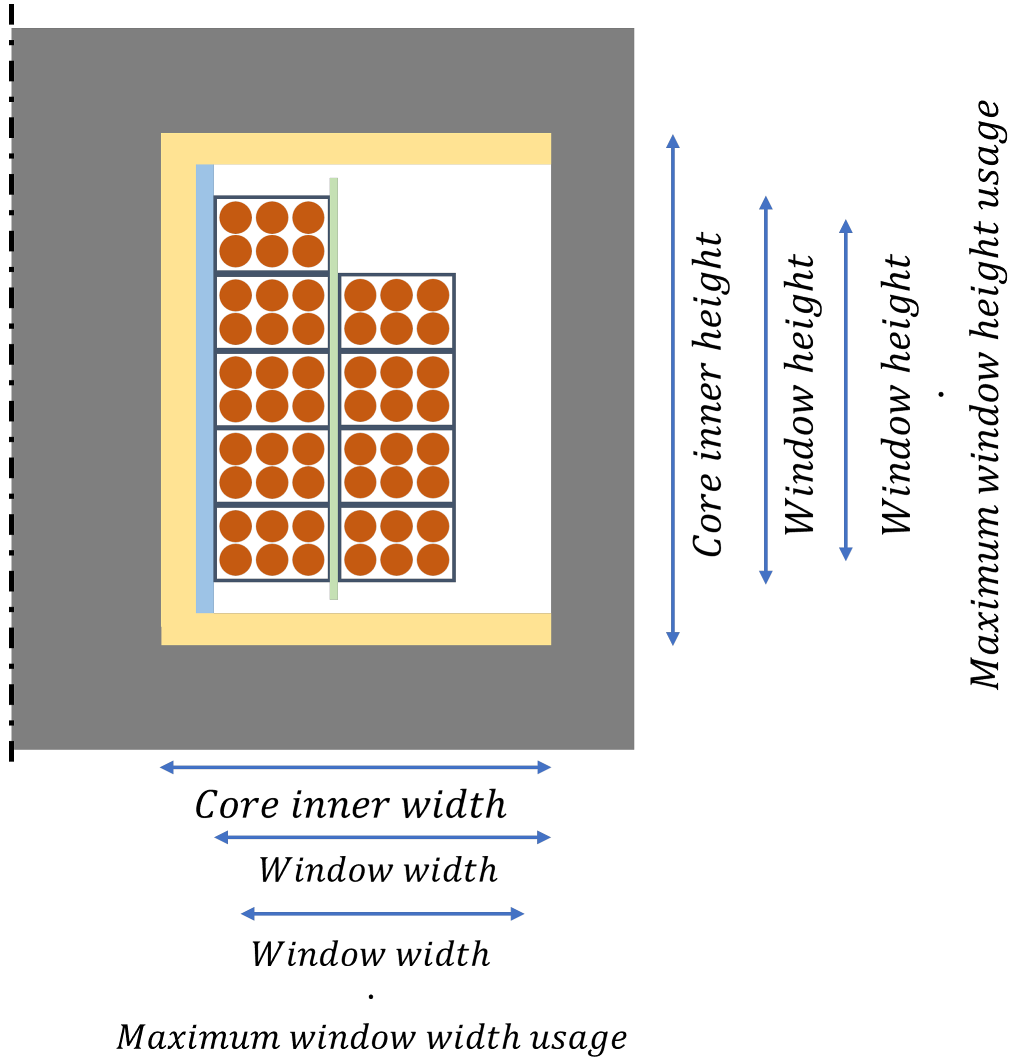
4.2.2.2 Window usage

One of the limitations for the winding is to fit the available window.

The window height and width are defined as the inner core height and width minus the space used by the bobbin:

$$Window\ height = Core\ inner\ height - 2 \cdot bobbin\ thickness$$
$$Window\ width = Core\ inner\ width - bobbin\ thickness$$

- The maximum available height is defined as the window height multiplied by the "Maximum window height usage" value:
  $$Available\ window\ height = Window\ height \cdot Maximum\ window\ height\ usage$$
- The maximum available width is defined as the window width multiplied by the "Maximum window width usage" value:
  $$Available\ window\ width = Window\ width \cdot Maximum\ window\ width\ usage$$



4.2.2.3 Design limits

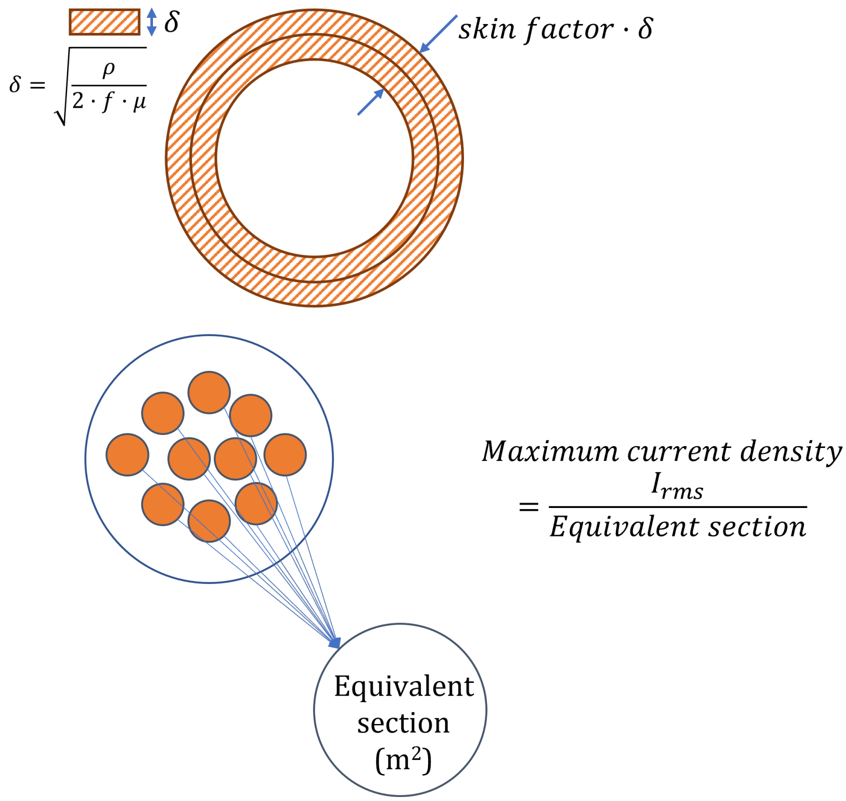
The use of different wires can be limited by two factors

- Skin depth
- Current density

If the "Limit skin depth" option is checked, only wires with a radius lower than 'Skin factor' times the skin depth at the fundamental frequency are used.

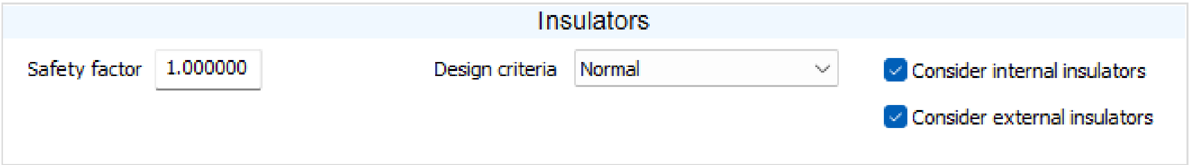
If the "Limit current density" option is checked, only wires that would have a current density lower than the limit are used. For this limit, the number of wires in parallel is taken into account as well.

For Litz wires or for wires in parallel, the current density is calculated as the total RMS current over the sections of every wire conducting in parallel.



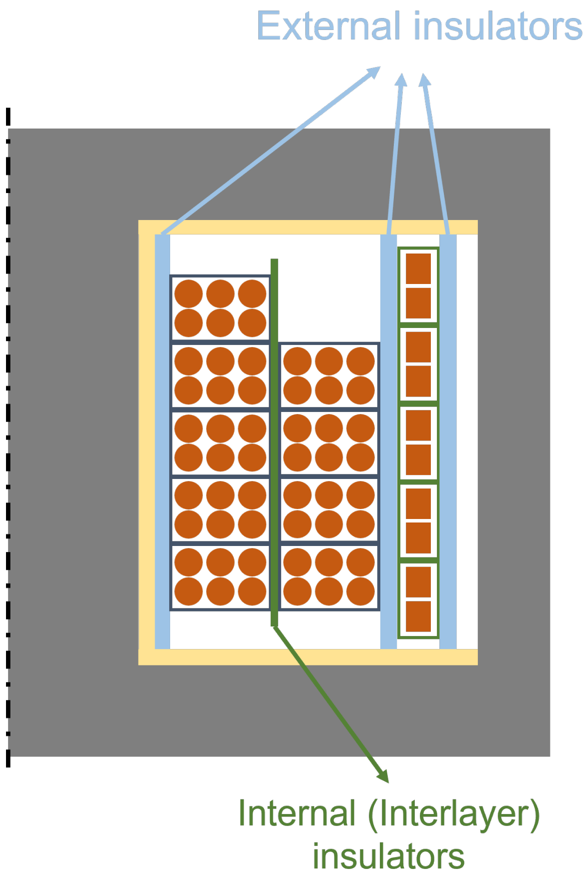
4.2.3 Insulators

This panel groups the options related to the insulators.



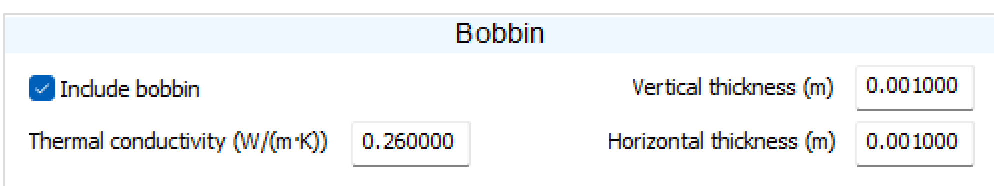
- **Design criteria**: chooses the criteria for the selection of the insulation. The available options are:
  - **Normal**: uses the thickness needed to block the required voltage, as given by the dielectric strength of the material.
  - **Avoid partial discharges**: uses the thickness needed to avoid partial discharges, as given by the manufacturer.
- **Safety factor**: factor by which the required insulation voltage is multiplied to increase insulation safety.
- **Consider internal insulators**: enable to take into account the possibility of adding insulation between different layers of the same winding (see image below).

- **Consider external insulators**: enable to take into account the possibility of adding insulation between different windings and between windings and core or ambient (see image below).

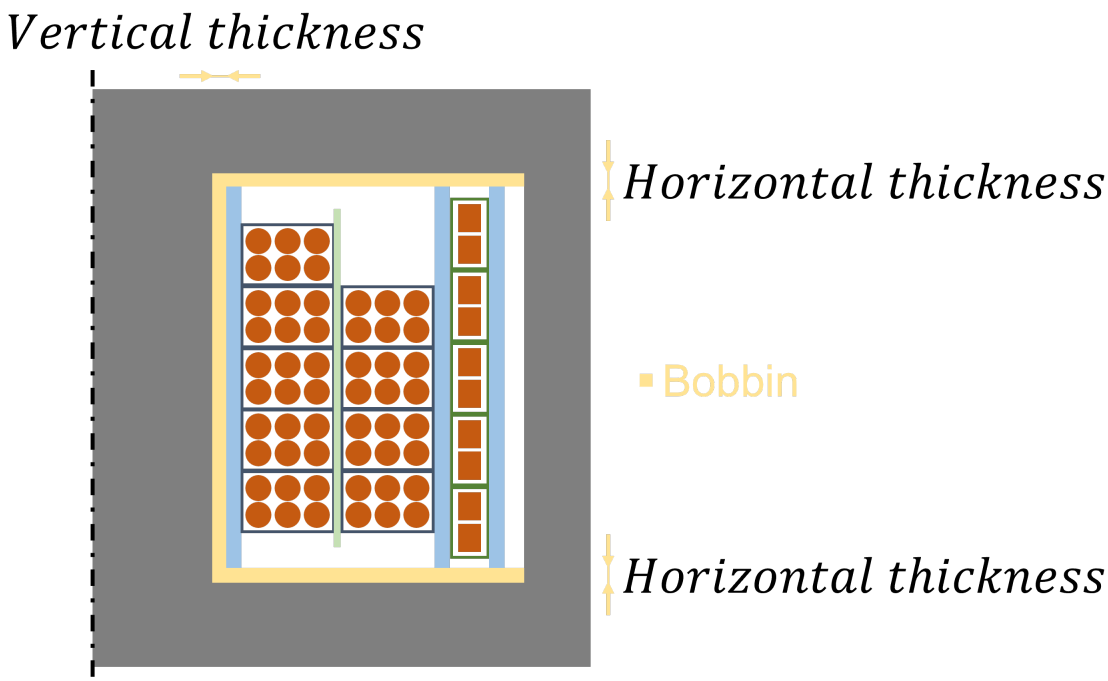


4.2.4 Bobbin

This panel groups the options related to the bobbin.



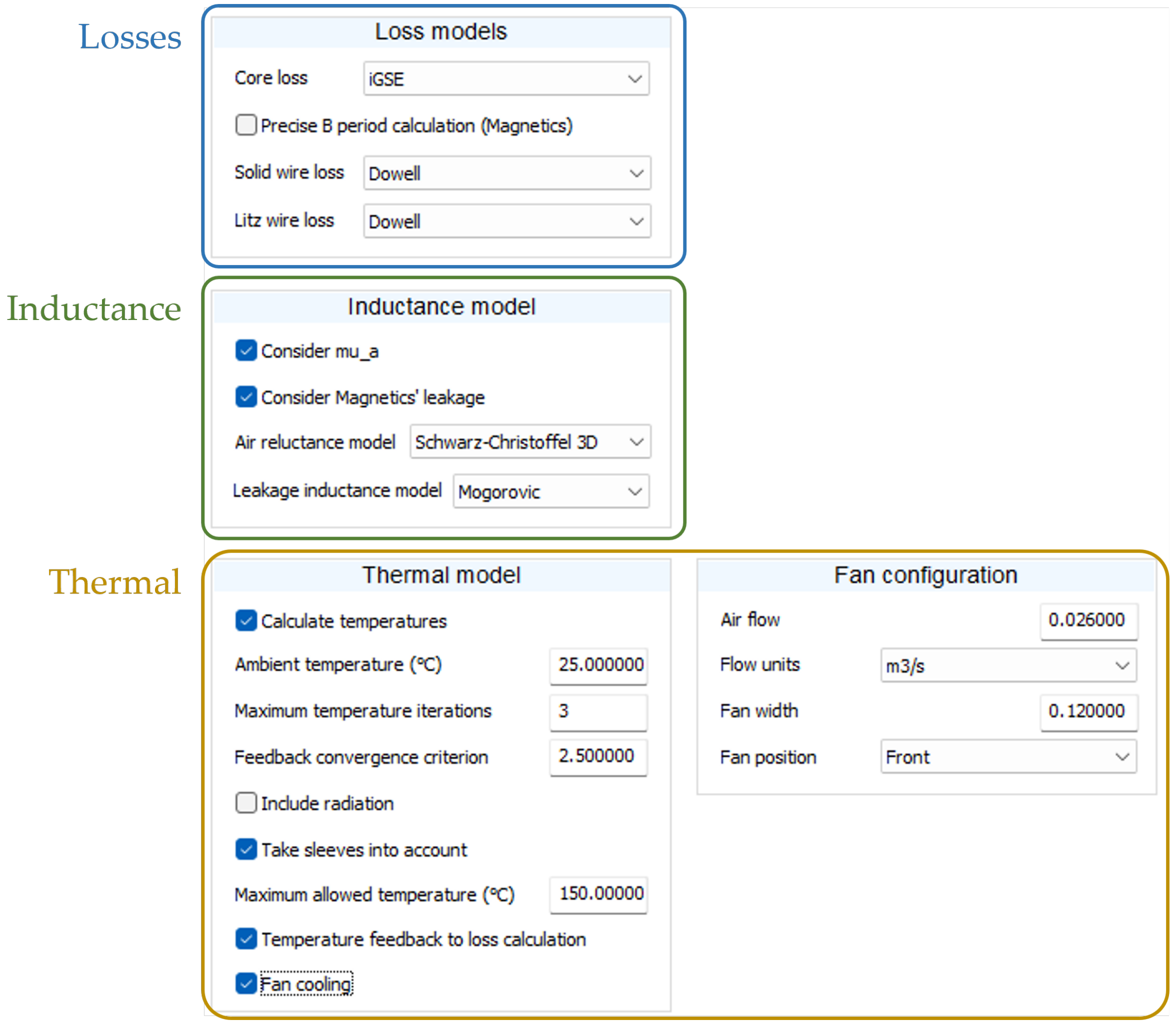
- **Include bobbin**: includes a bobbin in the design.
- **Thermal conductivity (W/(m·K))**: thermal conductivity of the bobbin material.
- **Vertical thickness (m)**: thickness, in meters, of the vertical walls of the bobbin, surrounding the central leg of the core (see image below).
- **Horizontal thickness (m)**: thickness, in meters, of the horizontal parts of the bobbin, parallel to the base (see image below).



4.3 Models

This tab allows the user to access every parameter related to the models used for the calculations.

It is divided in 3 panels:

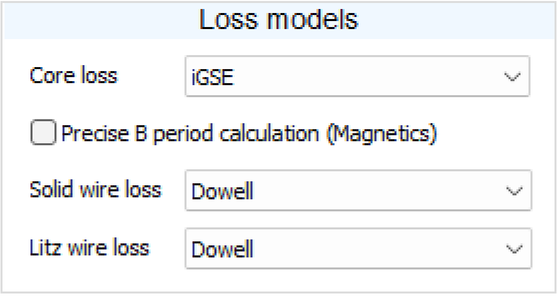


Every panel is described in its corresponding section:

- Losses
- Inductance
- Thermal

4.3.1 Losses

This panel groups the options related to the loss models.

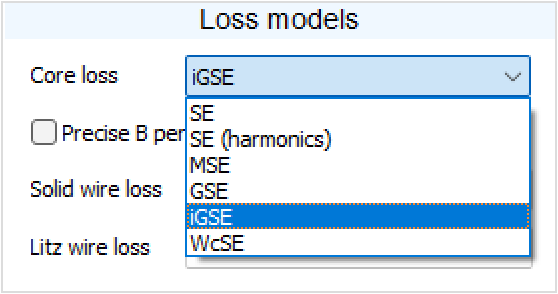


The different drop-down menus allow the selection of the model used for the calculation of:

- Core loss. The currently available options are:
  - **SE**: Steinmetz Equation applied to a single sinusoidal wave
  - **SE (harmonics)**: Steinmetz Equation applied to every harmonic
  - **MSE**: Modified Steinmetz Equation
  - **GSE**: Generalized Steinmetz Equation
  - **iGSE**: improved Generalized Steinmetz Equation
  - **WcSE**: Waveform-coefficient Steinmetz Equation
    The use of more complex models increases accuracy but also lengthens the design procedure. Currently, the model that provides the most accurate results (and longest design times) is iGSE and the one with a fastest design (and the lowest accuracy) is SE.
  - For the models that consider the full waveform, the option **Precise B period calculation (Magnetics)** allows a precise description of the B-H curve using the equation provided by Magnetics.
- Solid wire loss. The currently available options are:
  - **DC**: Considers only DC resistance
  - **Skin**: Considers DC resistance and Skin effect.
  - **Dowell**: Considers DC resistance, Skin effect and proximity effect.
- Litz wire loss. The currently available options are:
  - **DC**: Considers only DC resistance
  - **Skin**: Considers DC resistance and Skin effect.
  - **Dowell**: Considers DC resistance, Skin effect and proximity effect.
  - **Villar**: Considers DC resistance, Skin effect and proximity effect. Specifically applied to Litz wire.

4.3.1.1 Core

The core loss model to be used for the calculations can be selected through the corresponding drop-down menu:



The different models differ on accuracy and complexity:

SE: Steinmetz Equation

This is the most basic model, and is only well defined for pure sinusoidal waveforms (notice that for signals that are not sinusoidal, SmartNetics does the calculation using the total peak-to-peak value of flux density B, so results may not be accurate).

Reference: Steinmetz, C. P. (1892). On the law of hysteresis. Transactions of the American Institute of Electrical Engineers, 9(1), 1-64.

$$P_{c,SE} = V_c \cdot K_c \cdot f^{\alpha} \cdot \left(\frac{\Delta B_{pp}}{2}\right)^{\beta}$$

This model provides the fastest results, but presents some known issues:

- It does not take the full current/voltage waveform but only one harmonic
- It does not take into account B waveform, only amplitude
- It does not take into account the effects of the minor B loops due to the multiple 0 crossings of the voltage in a single period.

SE (harmonics): Steinmetz Equation applied to every harmonic

This model builds upon the Steinmetz Equation but applies its definition to every harmonic.

Reference: Steinmetz, C. P. (1892). On the law of hysteresis. Transactions of the American Institute of Electrical Engineers, 9(1), 1-64.

$$P_{c,SE} = V_c \cdot K_c \cdot f^{\alpha} \cdot \left(\frac{\Delta B_{pp}}{2}\right)^{\beta}$$

This model provides fast results, but presents some known issues:

- It does not take into account B waveform
- It does not take into account the effects of the minor B loops due to the multiple 0 crossings of the voltage in a single period.

MSE: Modified Steinmetz Equation

This model builds upon the Steinmetz Equation but takes into account not only the amplitude of the waveform, but also its shape, by means of the addition of an "Equivalent frequency".

Reference: Reinert, J., Brockmeyer, A., & De Doncker, R. W. (2001). Calculation of losses in ferro-and ferrimagnetic materials based on the modified Steinmetz equation. IEEE Transactions on Industry applications, 37(4), 1055-1061.

$$P_{c,MSE} = V_c \cdot f \cdot K_c \cdot f_{eq}^{\alpha-1} \cdot \left(\frac{\Delta B_{pp}}{2}\right)^{\beta}$$

This model provides a good trade off between speed and accuracy, but presents some known issues:

- It does not take into account the effects of the minor B loops due to the multiple 0 crossings of the voltage in a single period.

GSE: Generalized Steinmetz Equation

This model builds upon the Steinmetz Equation but takes into account not only the amplitude of the waveform, but also its shape, by including both B and dB/dt in the integral.

Reference: Li, J., Abdallah, T., & Sullivan, C. R. (2001, September). Improved calculation of core loss with nonsinusoidal waveforms. In Conference Record of the 2001 IEEE Industry Applications Conference. 36th IAS Annual Meeting (Cat. No. 01CH37248) (Vol. 4, pp. 2203-2210). IEEE.

$$P_{c,GSE} = V_c \cdot \frac{1}{T} k_i \int_0^T \left|\frac{dB}{dt}\right|^{\alpha} |B(t)|^{\beta-\alpha} dt$$

This model provides a good trade off between speed and accuracy, but presents some known issues:

- It can highly overestimate losses when there is a high fundamental-frequency amplitude with relatively small high frequency ripple.
- It does not take into account the effects of the minor B loops due to the multiple 0 crossings of the voltage in a single period.

iGSE: improved Generalized Steinmetz Equation

This model builds upon the previous one (GSE) but takes into account the effects of the minor B loops by dividing the full waveform in different loops that are treated separately.

Reference: Venkatachalam, K., Sullivan, C. R., Abdallah, T., & Tacca, H. (2002, June). Accurate prediction of ferrite core loss with nonsinusoidal waveforms using only Steinmetz parameters. In 2002 IEEE Workshop on Computers in Power Electronics, 2002. Proceedings. (pp. 36-41). IEEE.

$$P_{c,iGSE} = V_c \cdot \frac{1}{T} \int_0^T k_i \left|\frac{dB}{dt}\right|^{\alpha} \left(\Delta B_{pp}\right)^{\beta-\alpha} dt = V_c \cdot \frac{1}{T} k_i \left(\Delta B_{pp}\right)^{\beta-\alpha} \int_0^T \left|\frac{dB}{dt}\right|^{\alpha} dt$$

This model provides the best accuracy in exchange for the longest design

WcSE: Waveform-coefficient Steinmetz Equation

This model builds upon the Steinmetz Equation but takes into account not only the amplitude of the waveform, but also its shape, by including a factor that correlates different waveforms to a pure sinusoidal.

Reference: Shen, W., Wang, F., Boroyevich, D., & Tipton, C. W. (2008). Loss characterization and calculation of nanocrystalline cores for high-frequency magnetics applications. IEEE Transactions on Power Electronics, 23(1), 475-484.

$$P_{c,WcSE} = FWC \cdot V_c \cdot K_c \cdot f^{\alpha} \cdot \left(\frac{\Delta B_{pp}}{2}\right)^{\beta}$$

This model provides a good accuracy for some specific waveforms, like triangular or square, but:

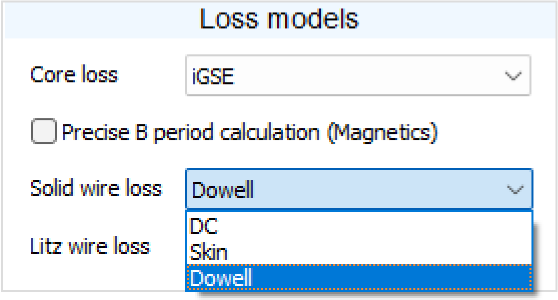
- It has as an increased error when the waveform differs from these
- It does not take into account the effects of the minor B loops due to the multiple 0 crossings of the voltage in a single period

In general, a more complex model provides a more accurate result, in exchange for a higher consumption of time and resources.

The increasing level of accuracy and resource consumption is depicted in the figure below. The selection of the appropriate model depends on the particular design needs and, for the best results, iGSE is recommended.

4.3.1.2 Solid wire

The solid wire loss model to be used for the calculations can be selected through the corresponding drop-down menu:



The different models differ on accuracy and complexity:

DC: Only DC resistance

This is the most basic model as it does not take into account the impact of frequency on resistance.

$$P_{w,DC} = I_{ef}^2 \cdot R_{DC}$$

This model provides the fastest results but presents some known issues:

- Does not take into account the effect of frequency on current distribution in a conductor.
- Does not take into account the effect adjacent wires on current distribution in a conductor.

Skin: DC resistance + Skin effect

This model adds to the previous one the impact of frequency on current distribution inside an isolated conductor.

The skin depth is defined by:

$$\delta_w = \sqrt{\frac{2\rho_w}{\omega \cdot \mu}} = \sqrt{\frac{2\rho_w}{2\pi f \cdot \mu}}$$

This model increases accuracy when the harmonic content at high frequencies is relevant but presents some known issues:

- Does not take into account the effect of frequency on current distribution in a conductor.

Dowell: DC resistance + Skin effect + Proximity effect

This model adds to the previous one the impact of frequency on current distribution inside a conductor when there are other conductors nearby.

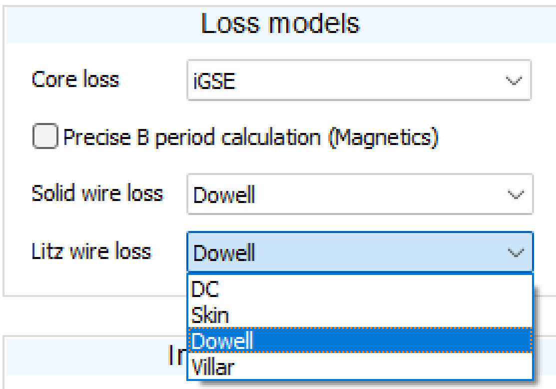
This model provides the most accurate results in exchange for an increased design time

In general, a more complex model provides a more accurate result, in exchange for a higher consumption of time and resources.

The increasing level of accuracy and resource consumption is depicted in the figure below. The selection of the appropriate model depends on the particular design needs and, for the best results, Dowell is recommended.

4.3.1.3 Litz wire

The Litz wire loss model to be used for the calculations can be selected through the corresponding drop-down menu:



DC: Only DC resistance

This is the most basic model and does not take into account the impact of frequency on resistance.

$$P_{w,DC} = I_{ef}^2 \cdot R_{DC}$$

This model provides the fastest results but presents some known issues:

- Does not take into account the effect of frequency on current distribution in a conductor.
- Does not take into account the effect adjacent wires on current distribution in a conductor.
- Does not take into account the twisting of Litz wire.

Skin: DC resistance + Skin effect

This model adds to the previous one the impact of frequency on current distribution inside an isolated conductor.

The skin depth is defined by:

$$\delta_w = \sqrt{\frac{2\rho_w}{\omega \cdot \mu}} = \sqrt{\frac{2\rho_w}{2\pi f \cdot \mu}}$$

This model increases accuracy when the harmonic content at high frequencies is relevant but presents some known issues:

- Does not take into account the effect of frequency on current distribution in a conductor.
- Does not take into account the twisting of Litz wire.

Dowell: DC resistance + Skin effect + Proximity effect

This model adds to the previous one the impact of frequency on current distribution inside a conductor when there are other conductors nearby.

This model provides very accurate results in exchange for an increase design time but presents some known issues:

- Does not take into account the twisting of Litz wire.

Villar: DC resistance + Skin effect + Proximity effect. Specific for Litz wire

This model builds on the previous one and increases its precision when applied to Litz wire.

Reference: Villar, I. (2010). Multiphysical characterization of medium-frequency power electronic transformers. PhD thesis.

$$P_{AC} = I_{AC,eff}^2 \cdot R_{DC} \cdot F_r$$

$$F_r = 1 + \frac{\gamma^4}{192}\left(\frac{1}{6} + \frac{\pi^2 n_s p_f}{4}\left(16m^2 - 1 + \frac{24}{\pi^2}\right)\right)$$

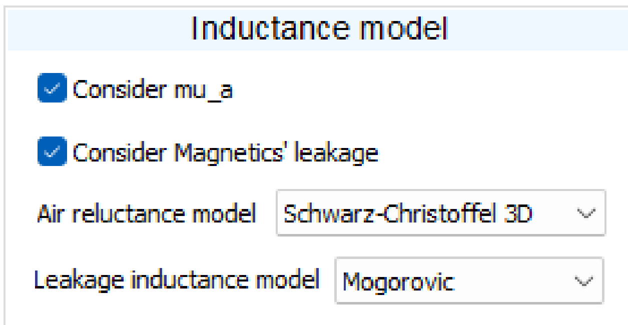
This model provides the most accurate results in exchange for an increase design time

In general, a more complex model provides a more accurate result, in exchange for a higher consumption of time and resources.

The increasing level of accuracy and resource consumption is depicted in the figure below. The selection of the appropriate model depends on the particular design needs and, for the best results, Litz approx is recommended.

4.3.2 Inductance

This panel groups the options related to the inductance model.



The fields are:

- **Consider mu_a**: takes into account the increase in permeability for increasing amplitudes of the signal. Changes mu_i for mu_a in the core reluctance calculation.
- **Consider Magnetics' leakage**: takes into account the additional inductance due to the flux through the air, as provided by Magnetics'. Only for Magnetics' materials.
- **Air reluctance model**: allows the selection of the model used for gap reluctance calculation. The currently available options are described in their corresponding section and can be selected through the corresponding drop-down menu:
  - Classical
  - Schwarz-Christoffel 2D
  - Schwarz-Christoffel 3D
- **Leakage inductance model**: allows the selection of the model used for reluctance calculation. The currently available options are described in their corresponding section and can be selected through the corresponding drop-down menu:
  - Dowell
  - Villar
  - Mogorovic

4.3.2.1 Air reluctance

Due to the relatively low reluctance of air, the flux lines in it take a much harder to predict path. That, in turn, makes its reluctance much harder to predict.

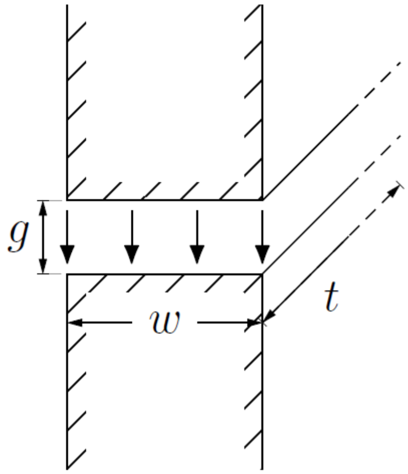
To account for that, different models have been proposed in the literature that allow a more precise calculation of the gap reluctance. 3 of those models are implemented in SmartNetics:

- Classical
- Schwarz-Christoffel 2D
- Schwarz-Christoffel 3D

The differences between the provided options are summarized here:

Classical

This model considers that the flux lines through the gap go from the top core to the bottom core in straight lines, crossing a section equal to that of the core, as depicted in the next figure.

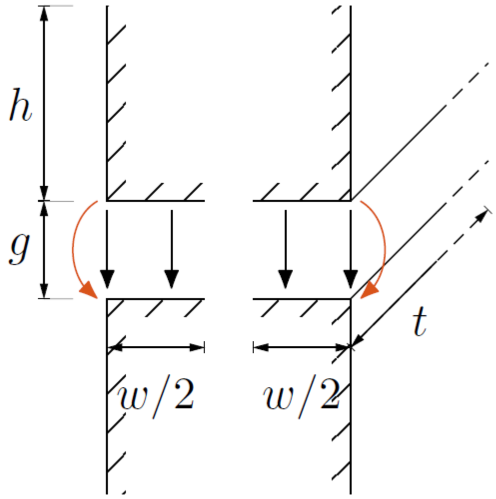


The reluctance due to the gap is defined by:

$$\mathcal{R}_g = \frac{g}{\mu_0 \cdot w \cdot t}$$

Schwarz-Christoffel 2D

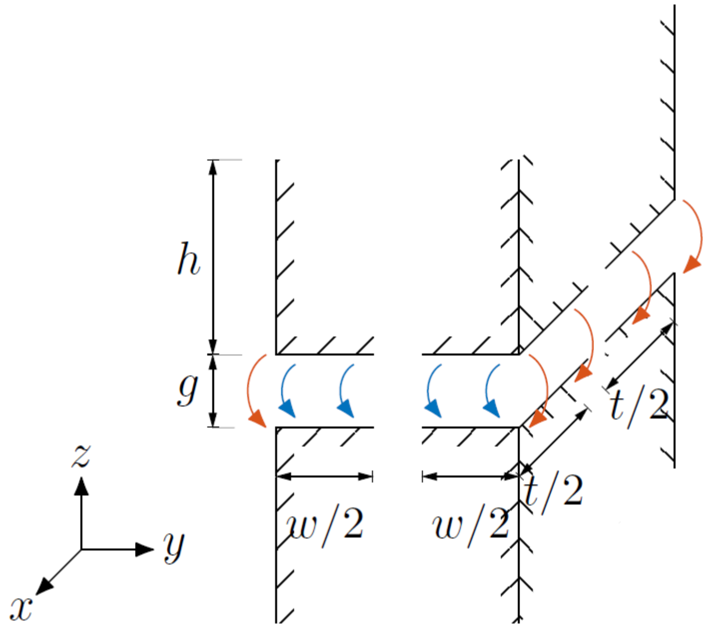
This model takes into account the increase in length and cross section that the path of the flux lines suffers due to fringing effect, considering 2 dimensions, as depicted in the next figure.



The reluctance expression considers several parameters that depend on geometry. The expression and the procedure can be accessed by the next link:
Reference: Balakrishnan, A., Joines, W. T., & Wilson, T. G. (1997). Air-gap reluctance and inductance calculations for magnetic circuits using a Schwarz-Christoffel transformation. IEEE Transactions on Power Electronics, 12(4), 654-663.

Schwarz-Christoffel 3D

This model takes into account the increase in length and cross section that the path of the flux lines suffers due to fringing effect, considering 3 dimensions, as depicted in the next figure.



The reluctance expression considers several parameters that depend on geometry. The expression and the procedure can be accessed by the next link:

Reference: Muhlethaler, J., Kolar, J. W., & Ecklebe, A. (2011, May). A novel approach for 3D air gap reluctance calculations. In 8th International Conference on Power Electronics-ECCE Asia (pp. 446-452). IEEE.

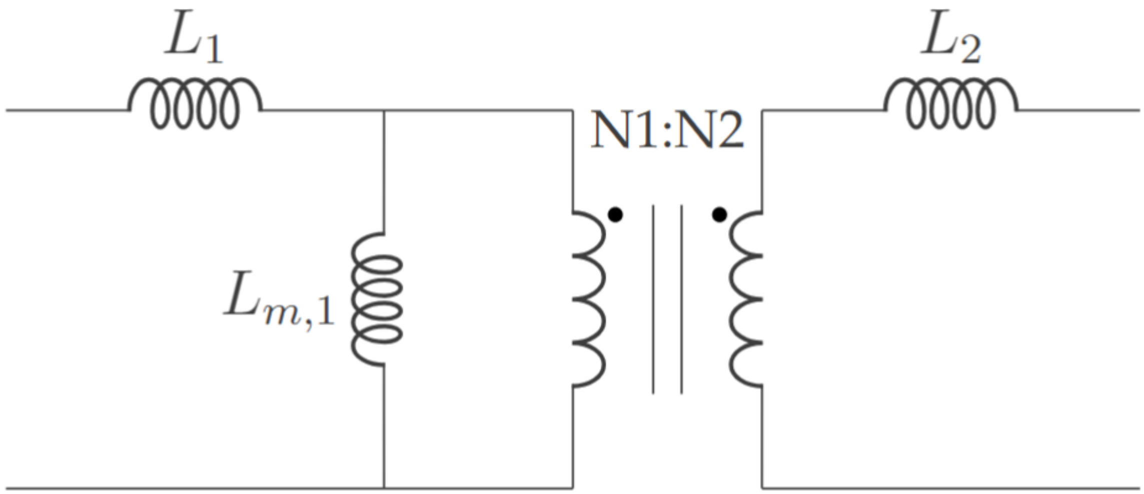
In general, a more complex model provides a more accurate result, in exchange for a higher consumption of time and resources.
The increasing level of accuracy and resource consumption is depicted in the figure below. The selection of the appropriate model depends on the particular design needs and, for the best results, Schwarz-Christoffel 3D (DC-3D) is recommended

4.3.2.2 Leakage inductance model

In an ideal, perfectly coupled transformer, all the magnetic flux generated by a winding is shared by the other.
In a real transformer though, most of the magnetic flux is shared by both windings but there is a portion of it that only relates to a single winding.

That impact of that flux that is not shared can be included in the equivalent circuit of the transformer by adding two inductances (on top of the magnetizing inductance), as shown in the next figure, where:

- $L_{m,1}$ is the magnetizing inductance referred to the primary winding.
- L_1 is the primary winding leakage inductance.
- L_2 is the secondary winding leakage inductance.



3 models are included in SmartNetics for the calculation of the leakage inductances:

- Dowell
- Villar
- Mogorovic

Dowell

The original model from which the others were derived. It considers the distances between wires of the same winding and the distance between windings.

Reference: Dowell, P. L. (1966, August). Effects of eddy currents in transformer windings. In Proceedings of the Institution of electrical Engineers (Vol. 113, No. 8, pp. 1387-1394). IEE.

The full procedure is described in the paper and in other references. The expression used to calculate leakage inductance is:

$$L_{lk} = N^2 \mu_0 \frac{MLT}{H_w} \left[\frac{d_{eq} \cdot m}{3} F_w + \frac{d_d}{2} + d_i \frac{(m-1)(2m-1)}{6m} \right]$$

Villar

A slightly different calculation of the inductance, following a deduction very similar to the original Dowell one, proposed by Irma Villar.

Reference: Villar, I. (2010). Multiphysical characterization of medium-frequency power electronic transformers. PhD thesis.

The expression for the calculation of leakage becomes:

$$L_{lk} = N^2 \mu_0 \frac{MLT}{H_w} \left[\frac{d_{eq} \cdot m}{3} F_w + \frac{d_d}{2} + d_i \frac{(m-1)}{2m} \right]$$

Mogorovic

A modification of Dowell's proposal, that may increase precision for low window fillings.

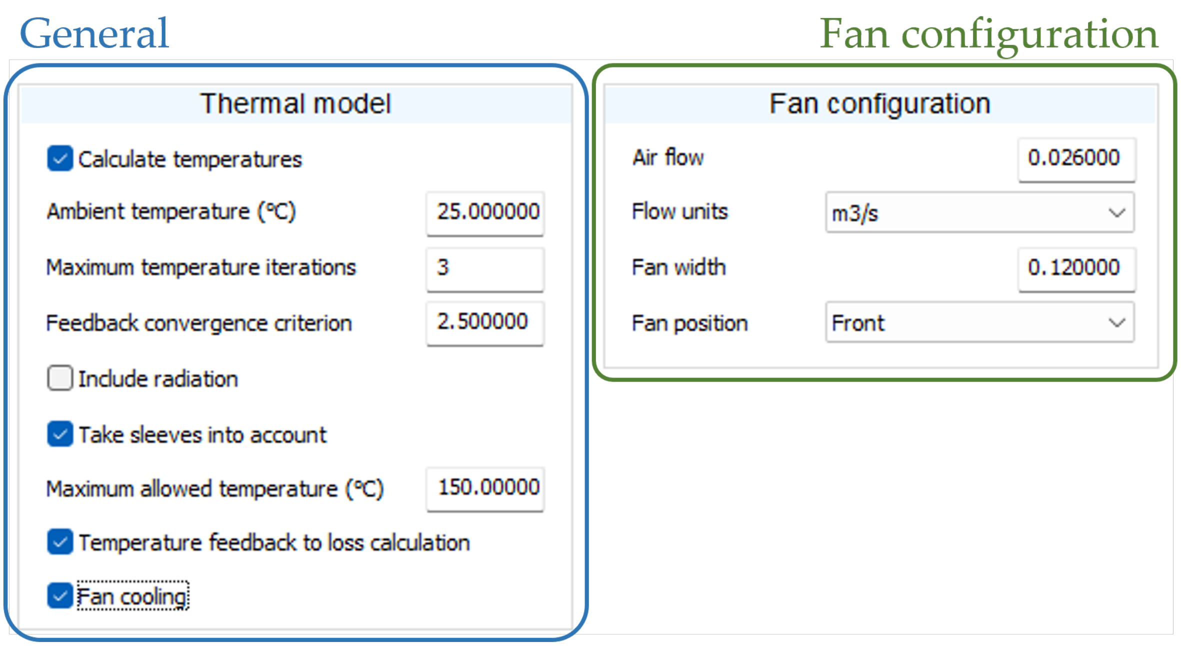
Reference: Mogorovic, M., & Dujic, D. (2017, October). Medium frequency transformer leakage inductance modeling and experimental verification. In 2017 IEEE Energy Conversion Congress and Exposition (ECCE) (pp. 419-424). IEEE.

It includes a correction factor to compensate the lower window filling:

$$K_R = 1 - \frac{1 - e^{-\pi h_w/(2(d_w + d_d/2)}}{\pi h_w/(2(d_w + d_d/2)} \}$$

4.3.3 Thermal

The options related to the temperature calculation are divided in two panels. The one on the left controls is always available and the one on the right only appears if the "Fan cooling" option is active.

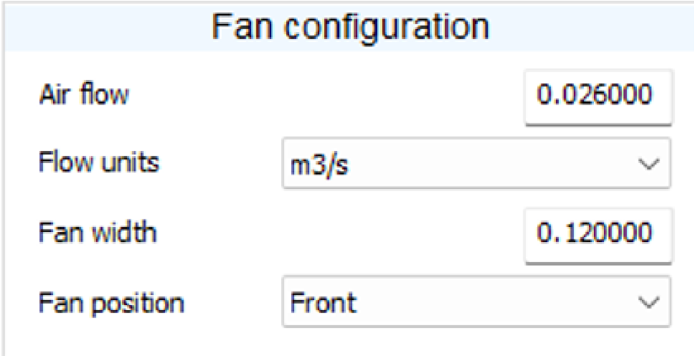


The available fields are:

- **Calculate temperatures**: select whether or not the temperature is calculated, taking into account ambient temperature and calculated losses. It enables the following fields:
- **Ambient temperature (°C)**: set ambient temperature, in degrees Celsius.
- **Maximum temperature iterations**: maximum number of iterations for the temperature calculation procedure. This parameter affects the calculation of wire losses, convection thermal resistance, and radiation thermal resistance, which depend on temperature.
- **Include radiation**: select whether or not to include radiation in the thermal resistance calculation. In general, activating this option is discouraged, since it can produce a temperature estimation much lower than expected. This model only takes into account the radiation emitted by the device, which reduces its temperature, but omits the radiation received, which would increase it, compensating the radiation impact.
- **Take sleeves into account**: select whether or not to take wire sleeves into account to increase accuracy.
- **Maximum allowed temperature (°C)**: select the maximum temperature at which designs are considered valid.
- **Temperature feedback to loss calculation**: select whether or not to consider the impact of temperature on losses. In the current version, only the impact on wire loss is considered (due to the change in conductor material resistance).
- **Fan cooling**: activating this option changes from natural to forced convection. If active, a new panel appears, with the "*Fan configuration*" options:
  - **Air flow** (the units depend on the next drop-down menu): volume per second (in m^3/s or cfm) or speed (in m/s) of air.
  - **Flow units**: units for the previous value: volume per second (in m^3/s or cfm) or speed (in m/s).
  - **Fan width**: width and height of the fan, assumed square.
    - If "Flow units" are set to m/s, this value has no impact on the design.
    - If "Flow units" are set to m^3/s or cfm, the speed of the air that cools the device is calculated as *Air flow/Fan area*.
  - **Fan position**: select the fan position, relative to the device. Available options are Front, Top and Bottom.

4.3.3.1 Fan configuration

This panel allows the configuration of the fan if the "Fan cooling" option is active in the thermal model configuration panel.

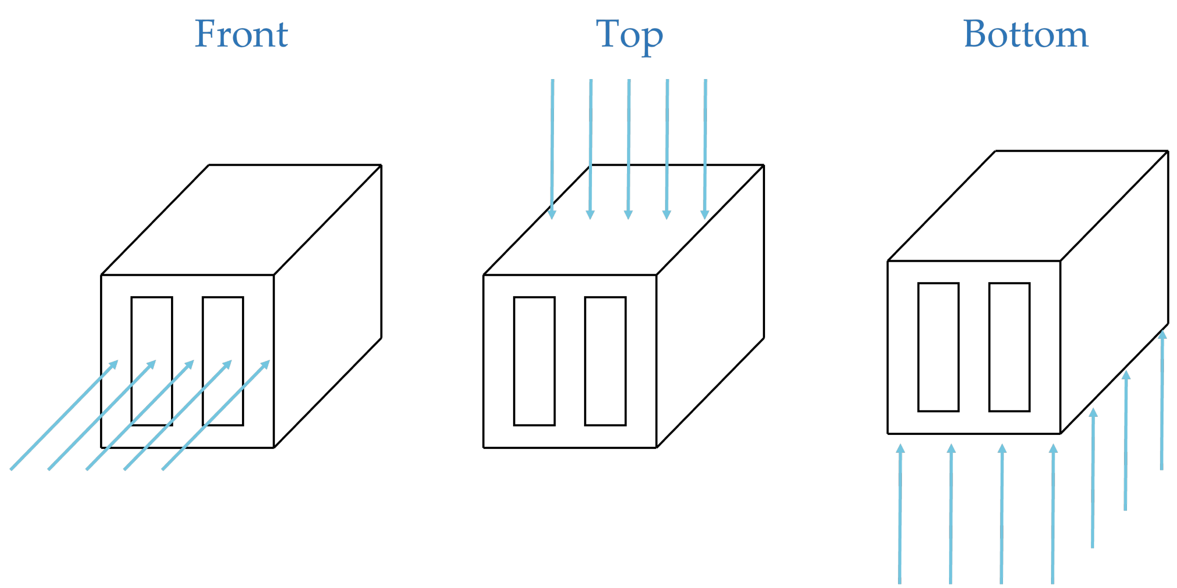


The 4 available configuration options are:

- **Air flow** (the units depend on the next drop-down menu): volume per second (in m^3/s or cfm) or speed (in m/s) of air.
- **Flow units**: units for the previous value: volume per second (in m^3/s or cfm) or speed (in m/s).
- **Fan width**: width and height of the fan, assumed square.
  - If "Flow units" are set to m/s, this value has no impact on the design.
  - If "Flow units" are set to m^3/s or cfm, the speed of the air that cools the device is calculated as:

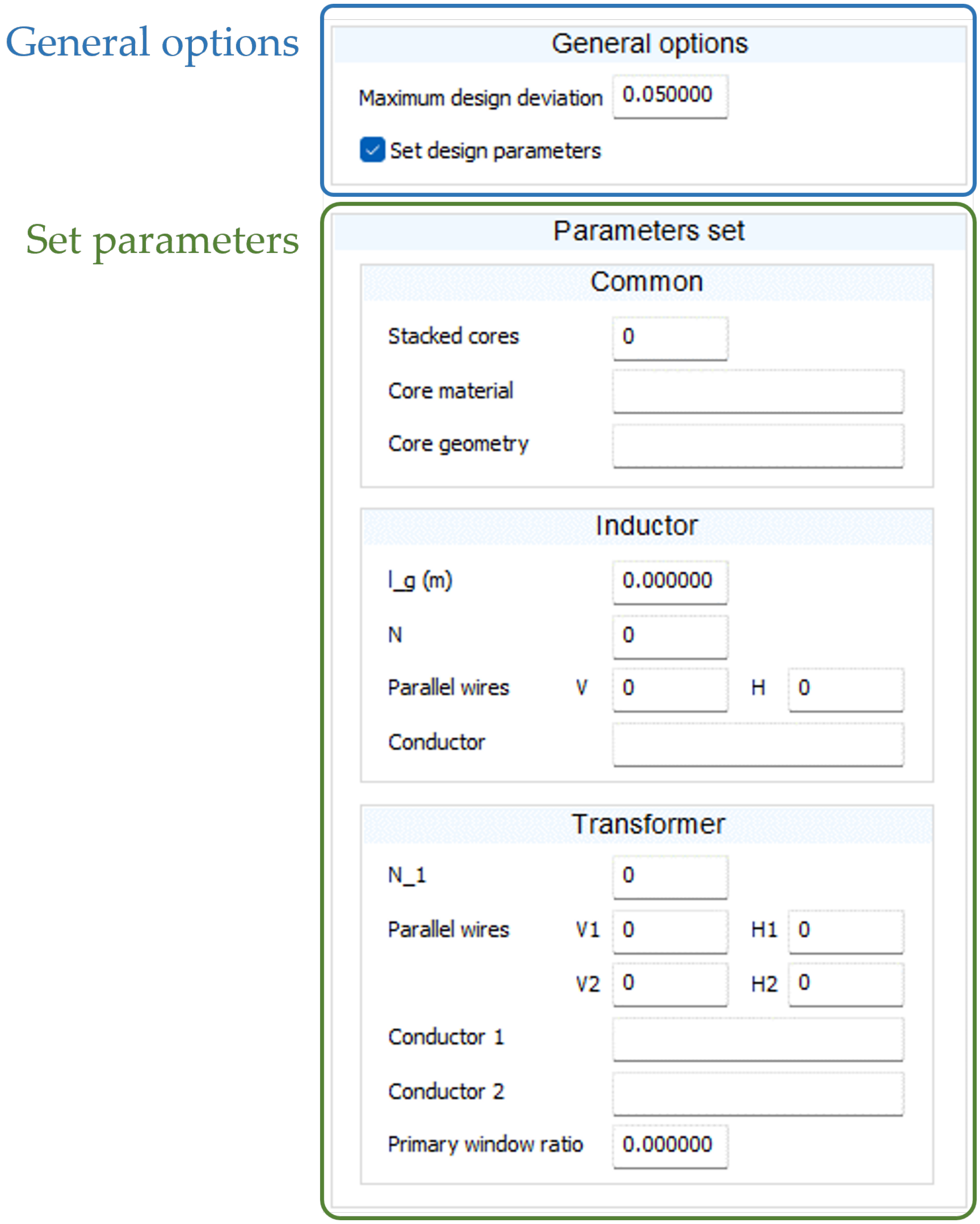
$$Air\ speed = \frac{Volume\ per\ time}{Area}$$

- **Fan position**: select the fan position, relative to the device. Available options are Front, Top and Bottom. See image below.



4.4 General

This tab allows the user to access the remaining general options for the configuration of the design procedure.
It is divided in 2 panels:



General options

- **Maximum design deviation**: set the maximum deviation of the imposed objective to regard a design as valid. Given as a per-unit value, the objective depends on the type of device:

- Inductor: maximum deviation from the imposed inductance. When using gapped cores a very tight margin can be used. For powder cores, a higher one (at least 5%) is suggested, since the inductance depends on the number of turns, which is discrete.
- Transformer: maximum deviation from the imposed turns ratio. For integer ratios (1, 2, 3 or 1, 0.5, 0.33) a very tight margin can be used. For irrational values or values with more decimal positions, a higher one (at least 5%) is suggested, since the ratio depends on the primary and secondary numbers of turns, which are discrete.

- **Fix design parameters**: Enables the option to set one or more parameters for the design (parameters left as 0 or empty strings remain unfixed and are calculated during the design procedure). Activating this option enables the **Fixed properties** panel.

*IMPORTANT NOTICE: Take into account that a fixed combination may not result in a valid design.

Fixed properties

- **Stacked cores**: fixes the number of cores stacked in parallel.
- **Core material**: fixes the core material. It has to be one of the materials active in the database.
- **Core geometry**: fixes the core geometry. It has to be one of the geometries active in the database.

For inductors:

- **l_g (m)**: fixes the total length of the path of the flux through air.
  - For an E core with standard gapping, this corresponds to 2 times that gap.
  - For cores with a single gap this corresponds to said gap.
  - For cores with distributed gapping this corresponds to the sum of every gap.
- **N**: fixes the number of turns.
- **V**: fixes the vertical number of wires in parallel. See the appropriate section for details.
- **H**: fixes the horizontal number of wires in parallel. See the appropriate section for details.
- **Conductor**: fixes the conductor. It has to be one of the conductors active in the database.

For transformers:

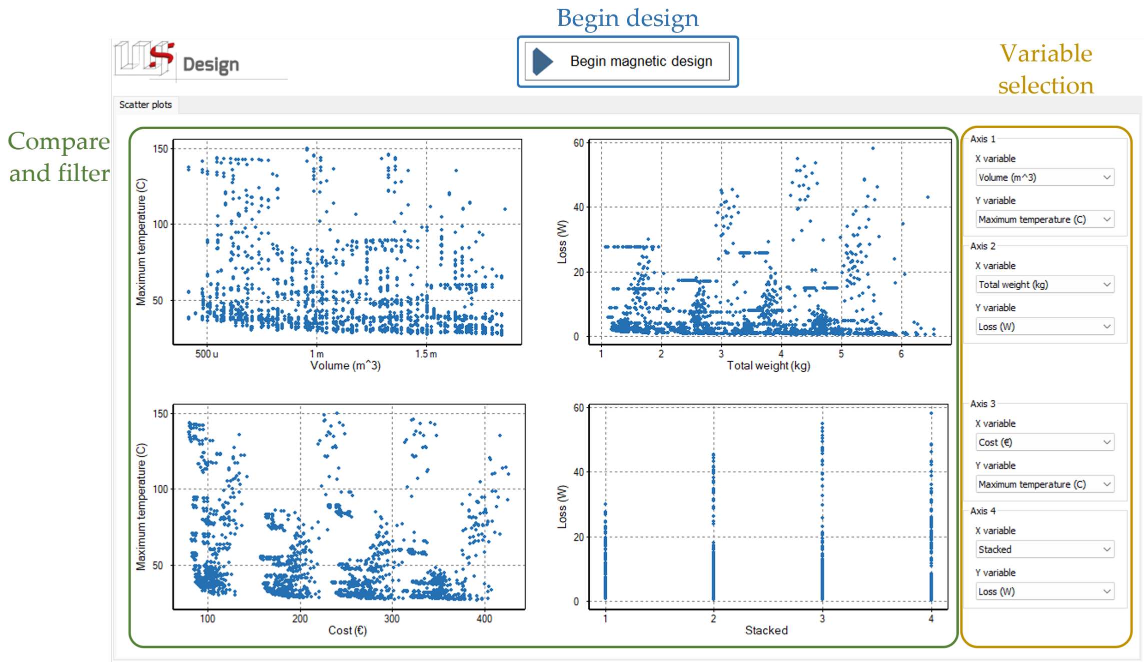
- **N_1**: fixes the number of turns for the primary winding. The primary winding is the one for which current and voltage have been defined.
- **V_1**: fixes the vertical number of wires in parallel for the primary winding. See the appropriate section for details.

- **H_1**: fixes the horizontal number of wires in parallel for the primary winding. See the appropriate section for details.
- **V_2**: fixes the vertical number of wires in parallel for the secondary winding. See the appropriate section for details.
- **H_2**: fixes the horizontal number of wires in parallel for the secondary winding. See the appropriate section for details.
- **Conductor 1**: fixes the conductor for the primary winding. It has to be one of the conductors active in the database.
- **Conductor 2**: fixes the conductor for the secondary winding. It has to be one of the conductors in the database.
- **Primary winding ratio**: fixes the per-unit maximum amount of window that the primary winding can occupy.

Design

5 Design

This dialog allows the user to start the design procedure and to graphically compare and filter out the designs.

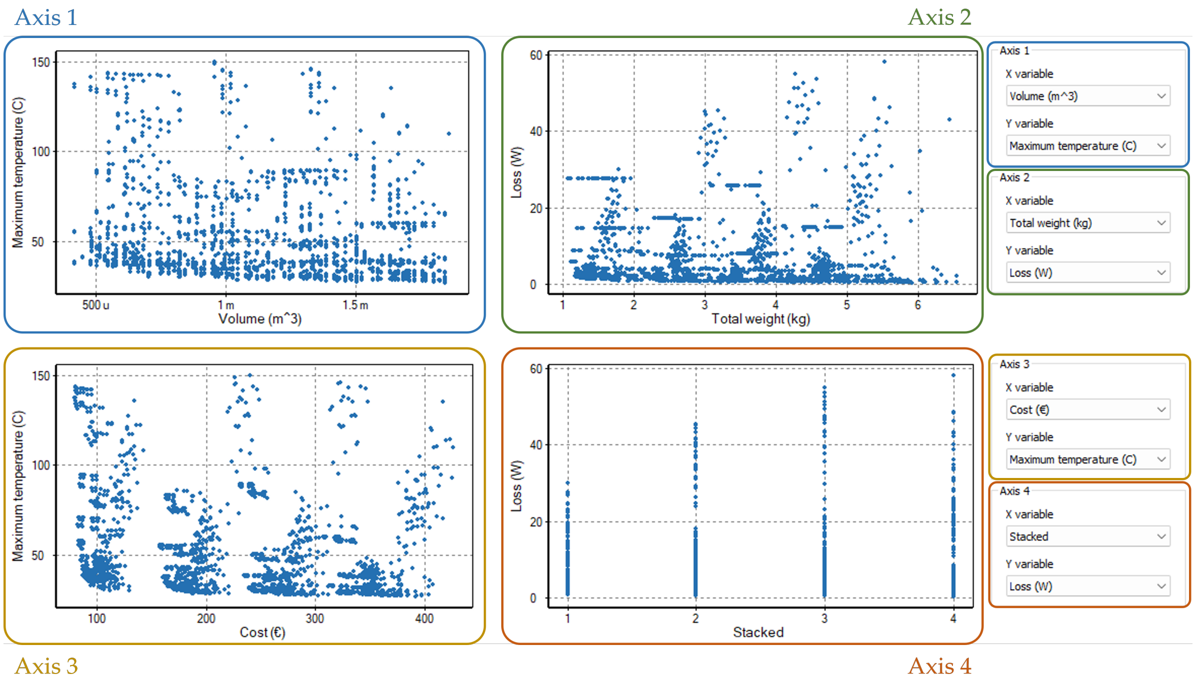


It is divided in three panels:

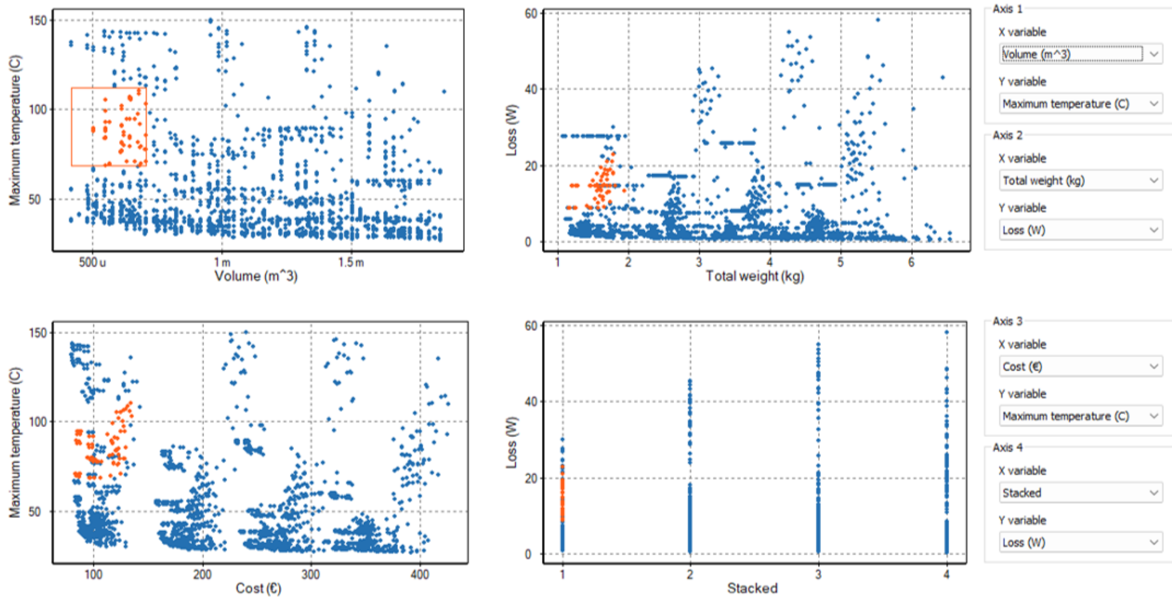
- **Begin design**: Begin the design procedure. Once it has begun, a wait-bar displays the number of valid designs already found and an estimation of the remaining time to try every possible design combination.
  The design can be stopped at any time and the already done designs are kept.
- **Compare and filter**: Graphically compare every design, and filter the ones that are considered candidates for the selection. Click and drag the cursor over any number of designs to select them in any graph. The designs selected in a graph area automatically displayed in the others. The devices shown in orange are the ones selected, and will be available in the next dialog: Selection.
- **Variable selection**: Select the variables to be used in every axis. For every axis, through the axes configuration panel, the user can select:
  - **X variable**: Variable to be used in the horizontal axis. The available options and their description are shown in the corresponding topic.
  - **Y variable**: Variable to be used in the vertical axis. The available options and their description are shown in the corresponding topic.

5.1 Compare and filter

The four axes shown have two variables each (X and Y), that can be selected by the drop-down menus on the right side. The available options and their description are shown in the corresponding topic.



Once the desired variables have been selected, the user can graphically compare every design, and filter the ones that are considered candidates for the selection. To do so, click and drag the cursor over any number of designs to select them in any graph.
By doing so, the designs selected in a graph area automatically displayed in the others, as shown in the next image:



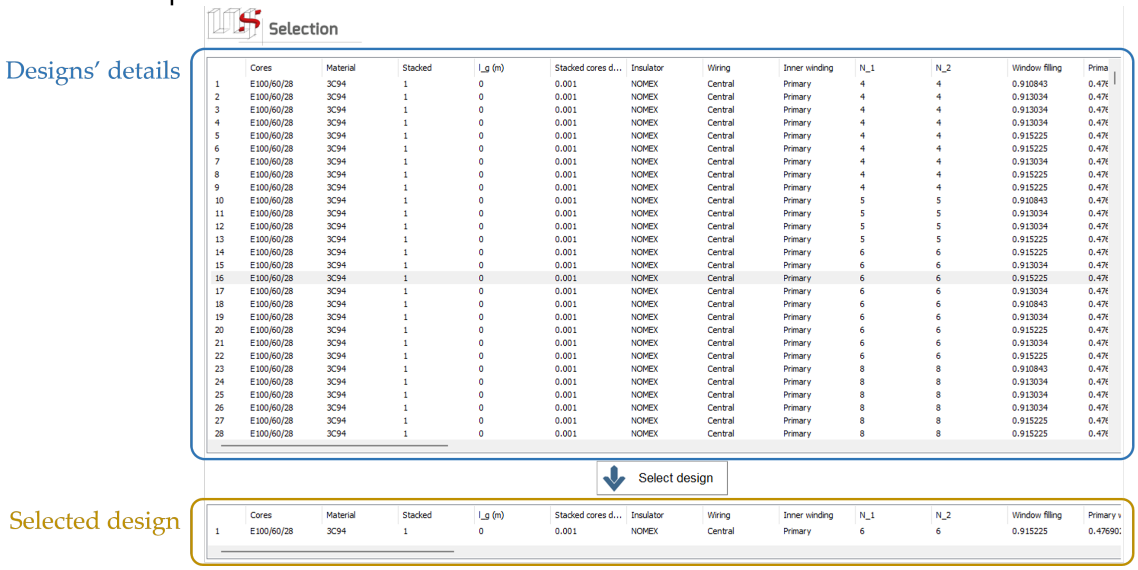
The devices shown in orange are the ones selected, and will be available in the next dialog: Selection.

Selection

6 Selection

This dialog allows the user to view every detail of every design selected in the previous step and to select a particular design for a deeper analysis in the next step.

It is divided in 2 panels:

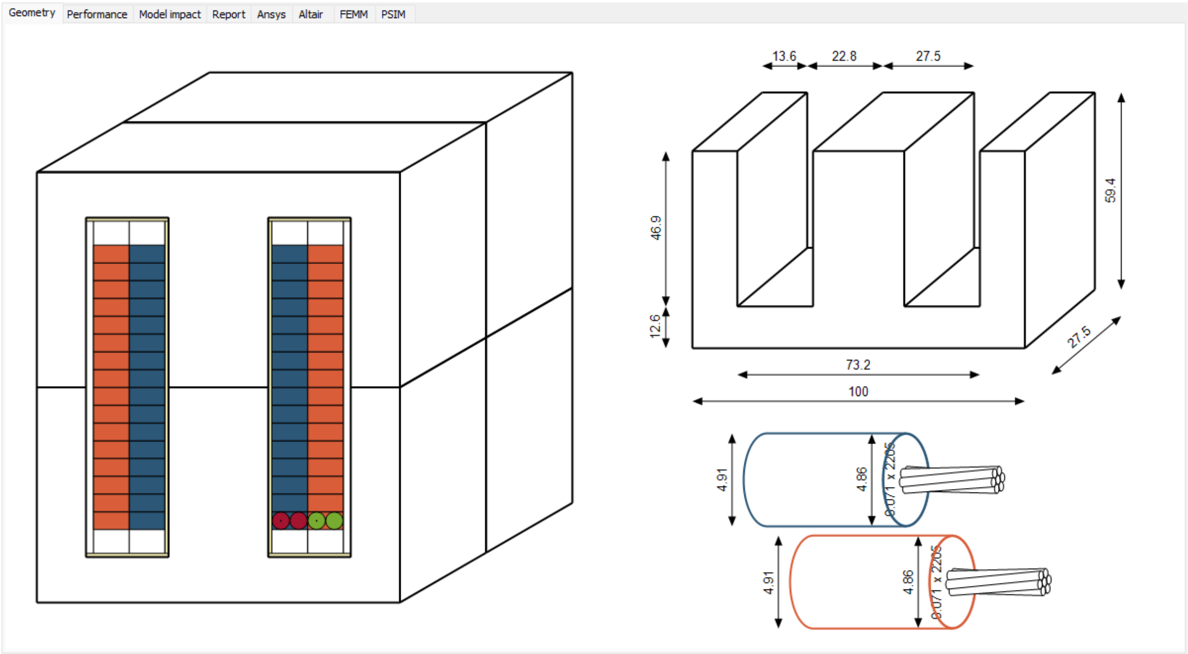


- **Designs' details**: here every detail of every design selected in the previous step is shown. To select a device pick any cell of said device.
- **Selected design**: shows the details of the device which analysis will be displayed in the next step. Selecting a device activates the button to go move to the last step: Device.

Device

7 Device

This dialog allows the user to analyze in detail the design selected in the previous step.



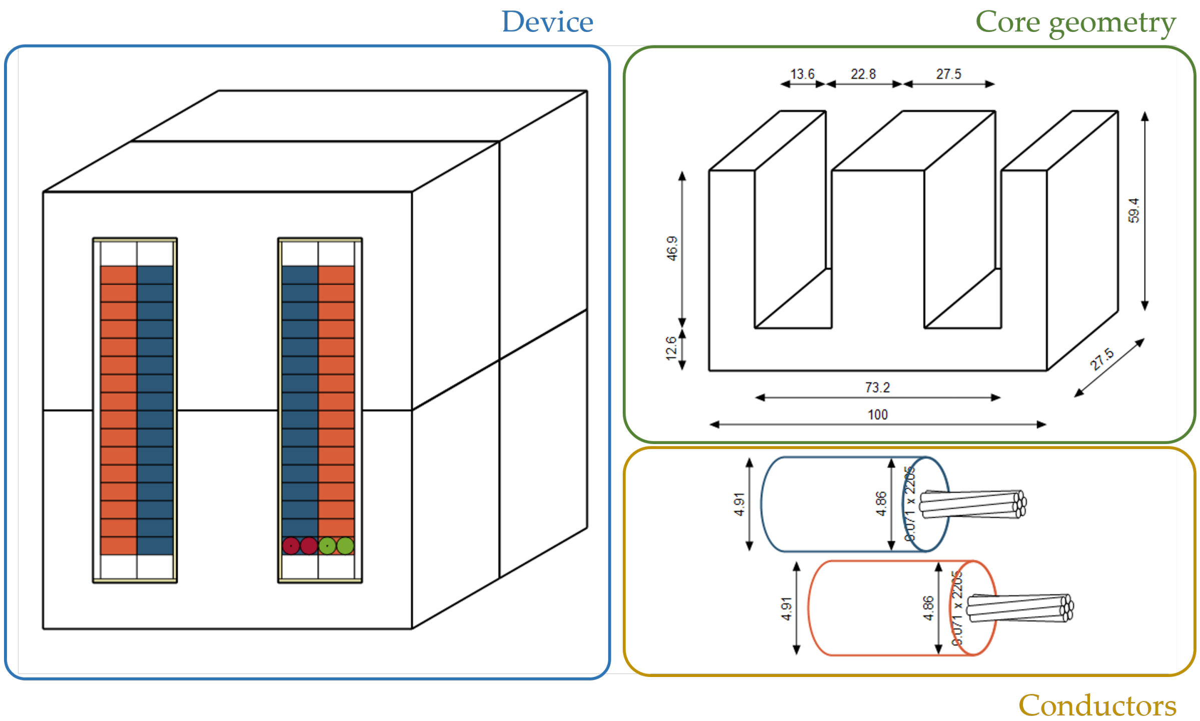
It is divided in 8 tabs:

- Geometry
- Performance
- Model impact
- Report
- Ansys
- Altair
- FEMM
- PSIM

7.1 Geometry

This tab shows the main dimensions of the device, the core and the conductors.

For an inductor, a single wire is shown. For a transformer, primary conductor is shown in blue, on top of the secondary conductor, shown in orange.

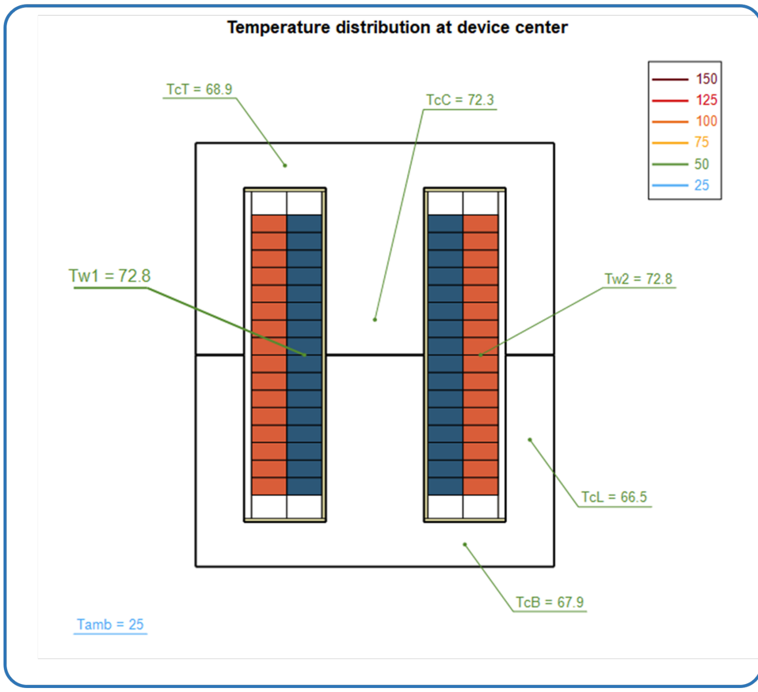


*The dimensions for core and conductor are in millimeters.

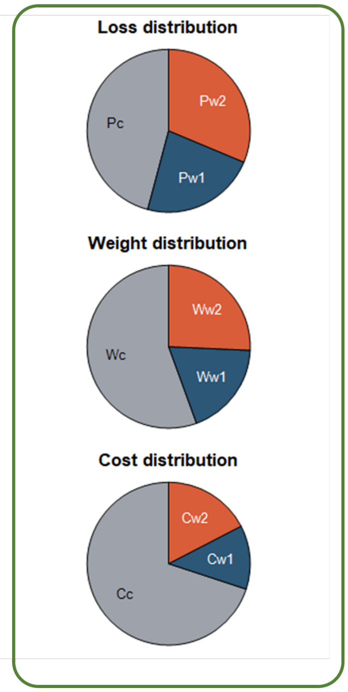
7.2 Performance

This tab shows the temperature distribution at the center of the device, and the distribution between core and windings of losses, weight and cost.

Temperature distribution



Core-windings distributions



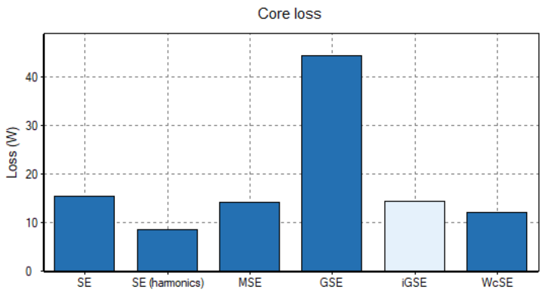
- **Temperature distribution**: The calculated temperatures (in °C) are:
  - Tamb: Ambient temperature
  - TcC: Temperature at the center of the central leg of the core
  - TcL: Temperature at the center of the lateral legs of the core
  - TcB: Temperature at the center of the bottom yoke of the core
  - TcT: Temperature at the center of the top yoke of the core
  - Tw (only for inductors): Winding temperature
  - Tw1 (only for transformers): Primary winding temperature
  - Tw2 (only for transformers): Secondary winding temperature

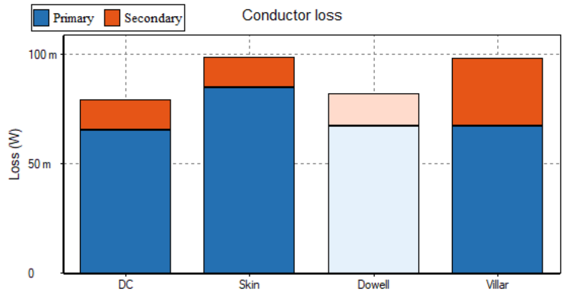
- **Core-windings distributions**:
  - Loss distribution: Relative impact of: Core loss (Pc), Winding loss (Pw, only for inductors), Primary winding loss (Pw1, only for transformers) and secondary winding loss (Pw2, only for transformers).
  - Weight distribution: Relative impact of: Core weight (Wc), Winding weight (Ww, only for inductors), Primary winding weight (Ww1, only for transformers) and secondary winding weight (Ww2, only for transformers).
  - Cost distribution: Relative impact of: Core cost (Cc), Winding cost (Cw, only for inductors), Primary winding cost (Cw1, only for transformers) and secondary winding cost (Cw2, only for transformers).

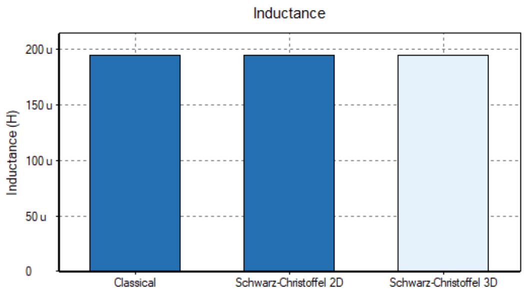
7.3 Model Impact

This tab shows the impact of the selected models on the calculations.

For the differences between the models, please refer to the sections related to core loss, solid wire loss, litz wire loss and inductance.

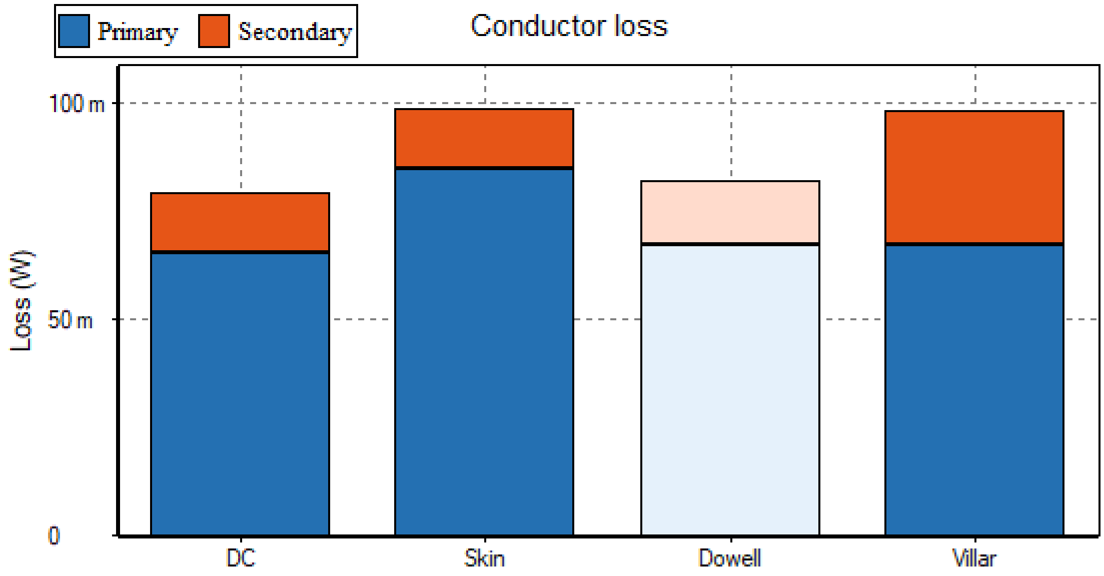






The model selected during the configuration step, which is the one that was used for the design is shown in lighter color, while the results if the other models are applied to the same device are shown in solid colors.

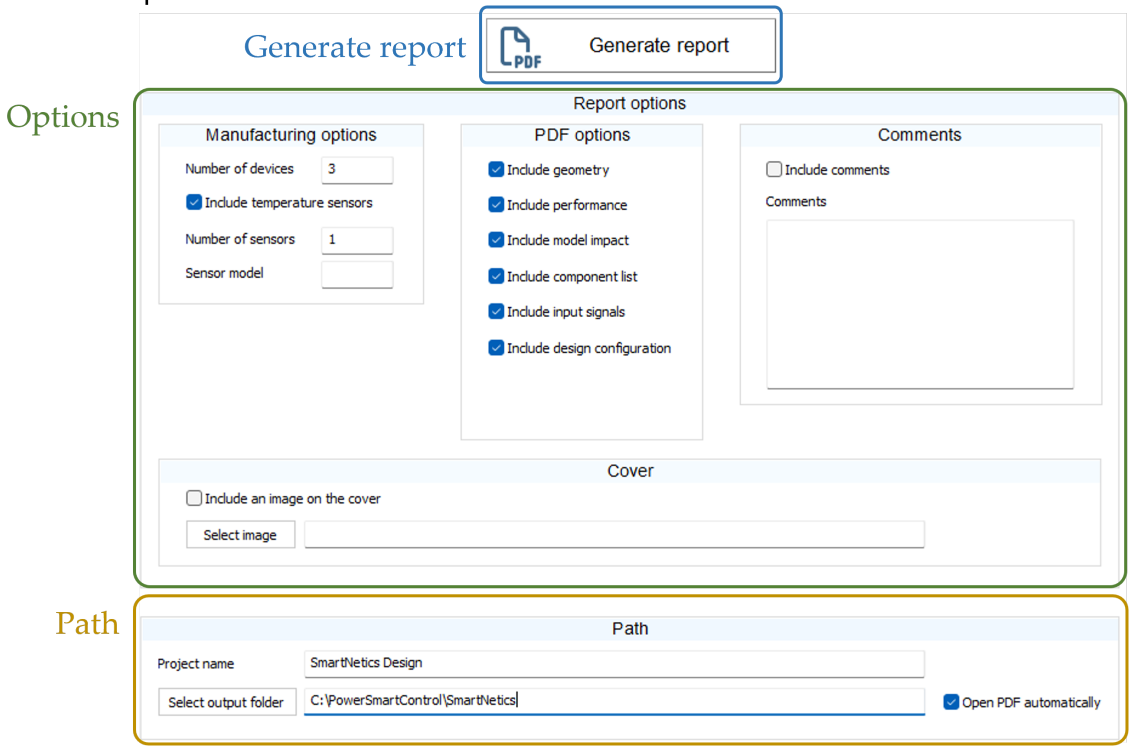
*For the conductor loss comparison, when designing a transformer, primary and secondary windings of the models that are not in use are also differentiated by their colors:



7.4 Report

This tab allows the user to generate a PDF report with the description of the selected device.

It consists of 3 panels:



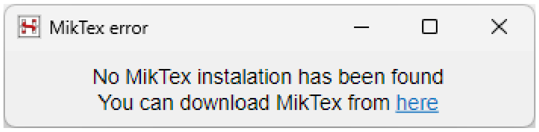
- **Generate report**: generate a PDF report with the configuration selected below. A LaTeX compiler is needed for the generation.

- **Options**: select the options for the PDF report generation. The available options are:
  - **Manufacturing options**
    - **Number of devices**: number of devices to be manufactured. If more than one is selected, additional tables are generated with the number of components to buy for the manufacture of a single unit and the total number of units.
    - **Include thermocouples**: select whether to specify to the manufacturer that thermocouples need are to be added. It enables the next fields.
      - **Thermocouple number**: number of thermocouples per device.
      - **Thermocouple model**: name of the thermocouple model to be used.
  - **PDF options**. select what pages are to be included in the PDF report.
    - **Include geometry**: include the 3D views shown in the Geometry tab.
    - **Include performance**: include the performance plots shown in the Performance tab.
    - **Include model impact**: include the model impact comparison plots shown in the Model impact tab.
    - **Include component list**: include a table of the components need to build one device. In case more than one is selected, an additional table is generated for the total amount of components.
    - **Include input signals**: includes a plot of the input signals used. For inductors only current is plotted while for transformers the voltage is shown as well.
    - **Include design configuration**: includes a text report of the options selected in the Configuration dialog.
  - **Comments**
    - **Include comments**: select whether or not to include a comments page. The comments can be added by the user in the corresponding text box.
  - **Cover**: configure the PDF cover
    - **Include an image on the cover**: select whether you want to include your own image on the cover or not.
    - **Select image**: browse your computer for an image to be included in the cover, alongside SmartNetics logo.

- **Path**: configure the options regarding the path of the PDF file.
  - **Project name**: name to be displayed in every page of the file and in the cover page.
  - **Select output folder**: select where to generate the PDF file. The user has to have access to the selected folder and privileges to create files there.
  - **Open PDF automatically**: select whether to automatically open the PDF once it is created.

7.4.1 LaTeX compiler

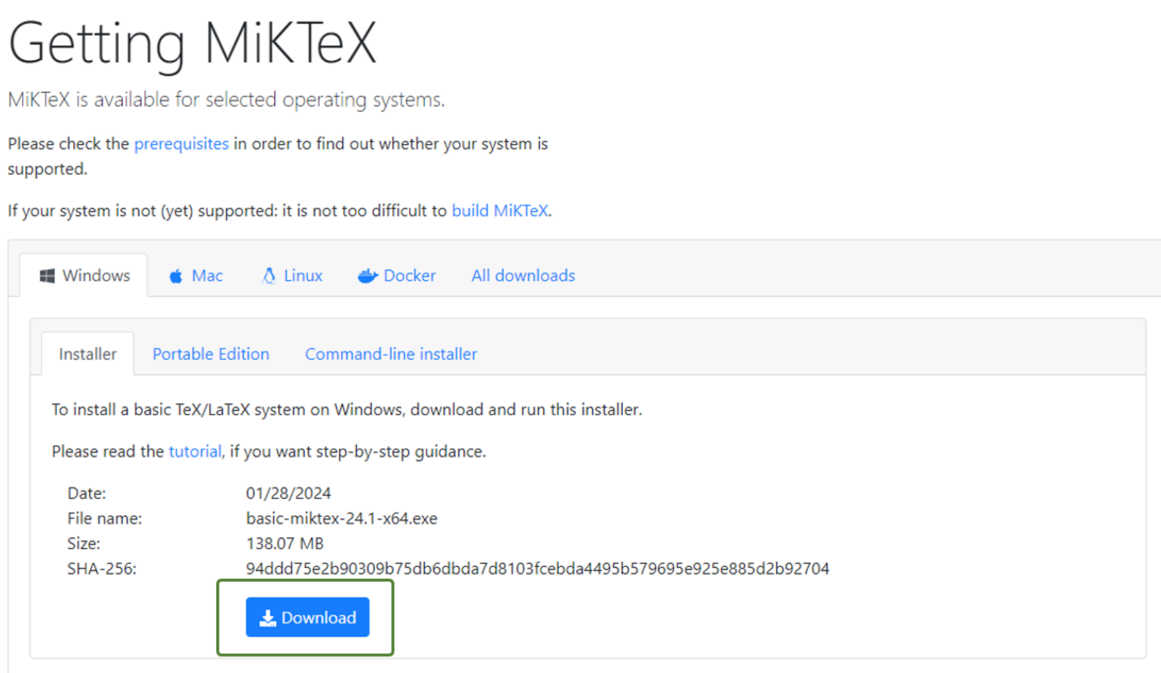
The PDF report is generated by compiling a LaTeX file using MikTeX (which is a third party free compiler).
In case no MikTeX installation is found, a message will be displayed showing the link for its free download:



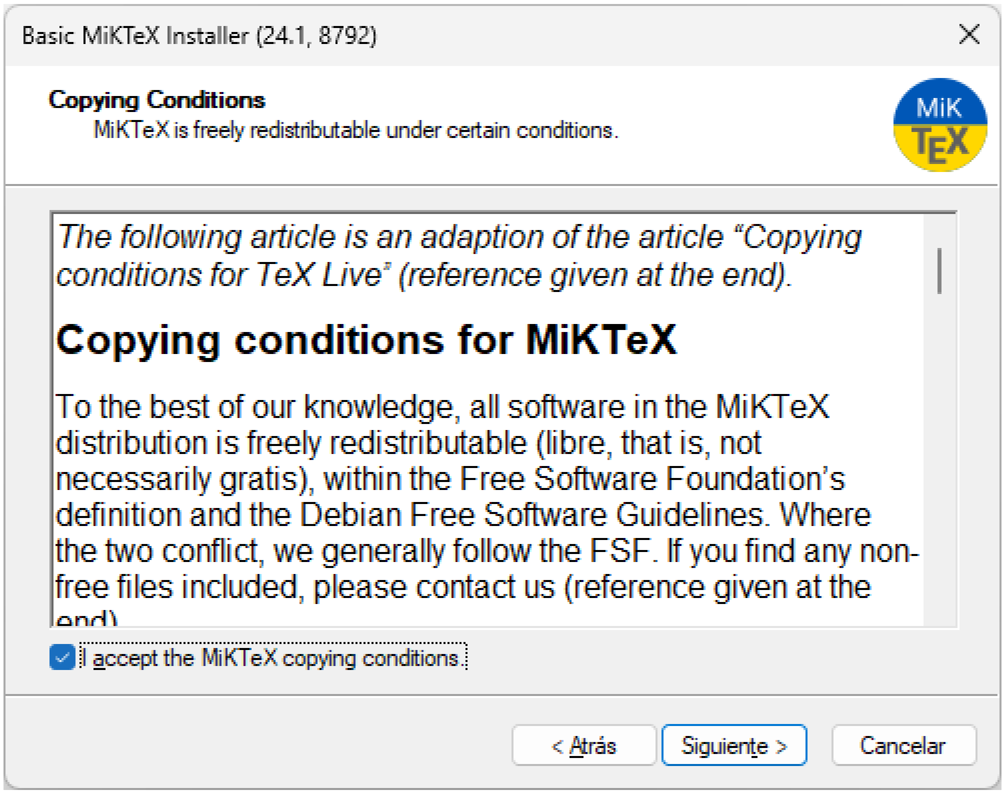
Upon clicking the link, the MikTeX web will open and the next steps can be followed for its installation.

MikTeX installation

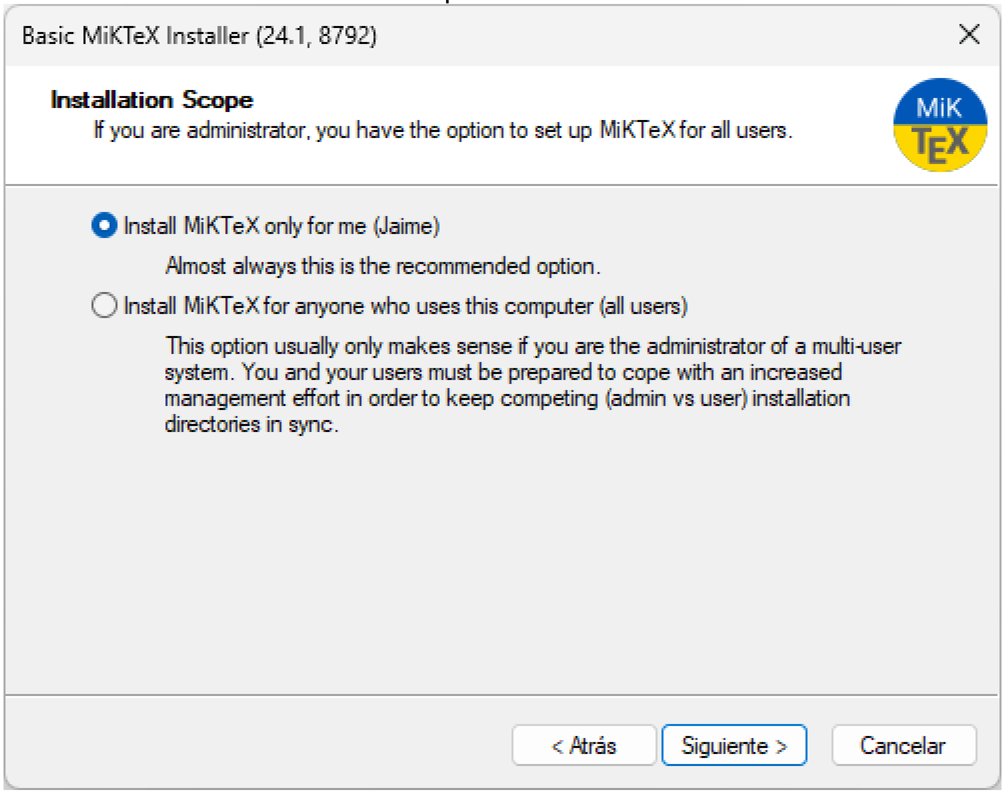
MikTeX can be downloaded from its web and clicking in the button shown below:



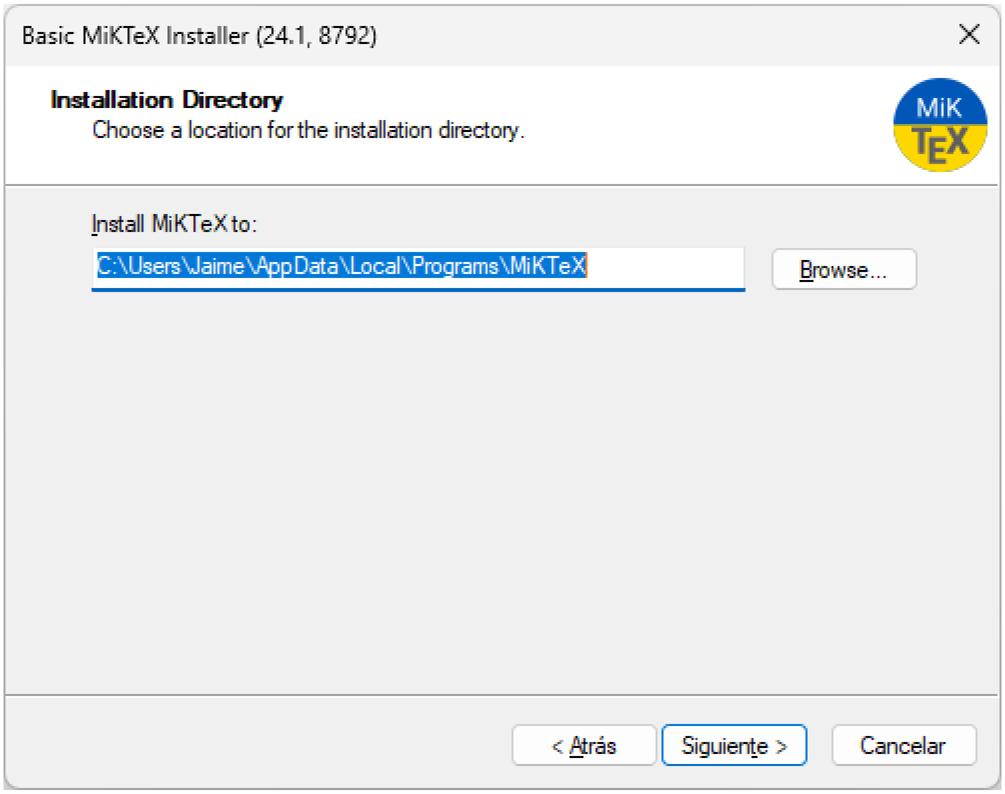
Once the installer is launched, the user needs to accept the copying conditions to carry on the installation:



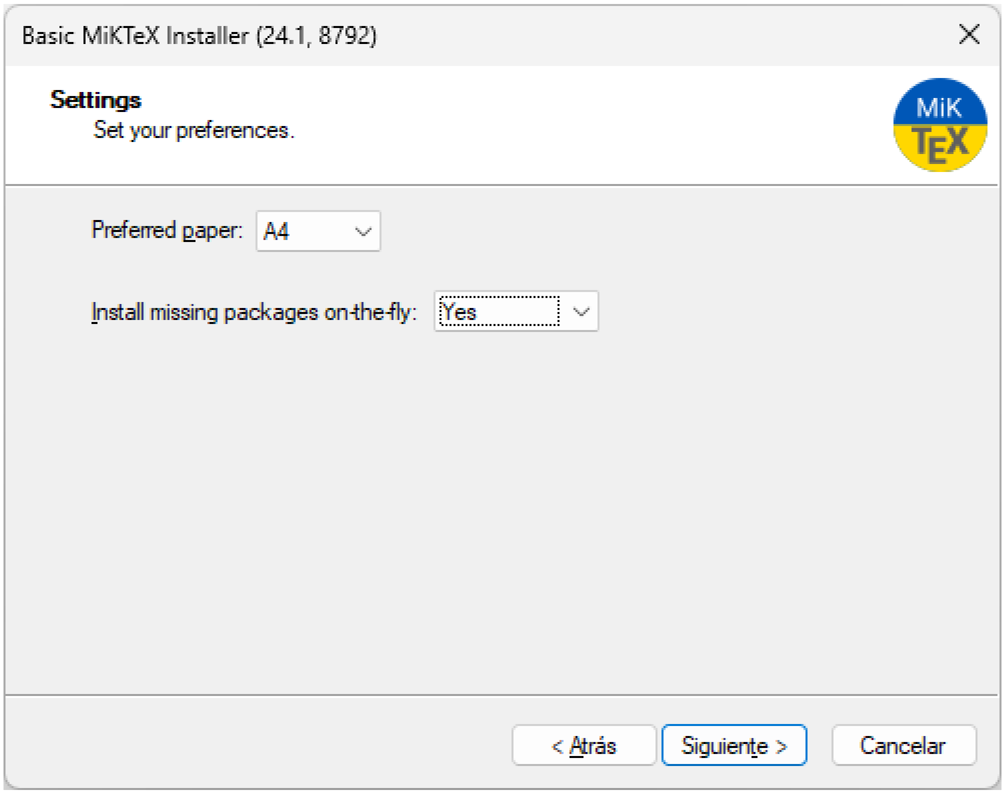
If the user accepts, the next step is to select whether to install it for every user or only the current one. The decision is up to the user and has no effect on SmartNetics:



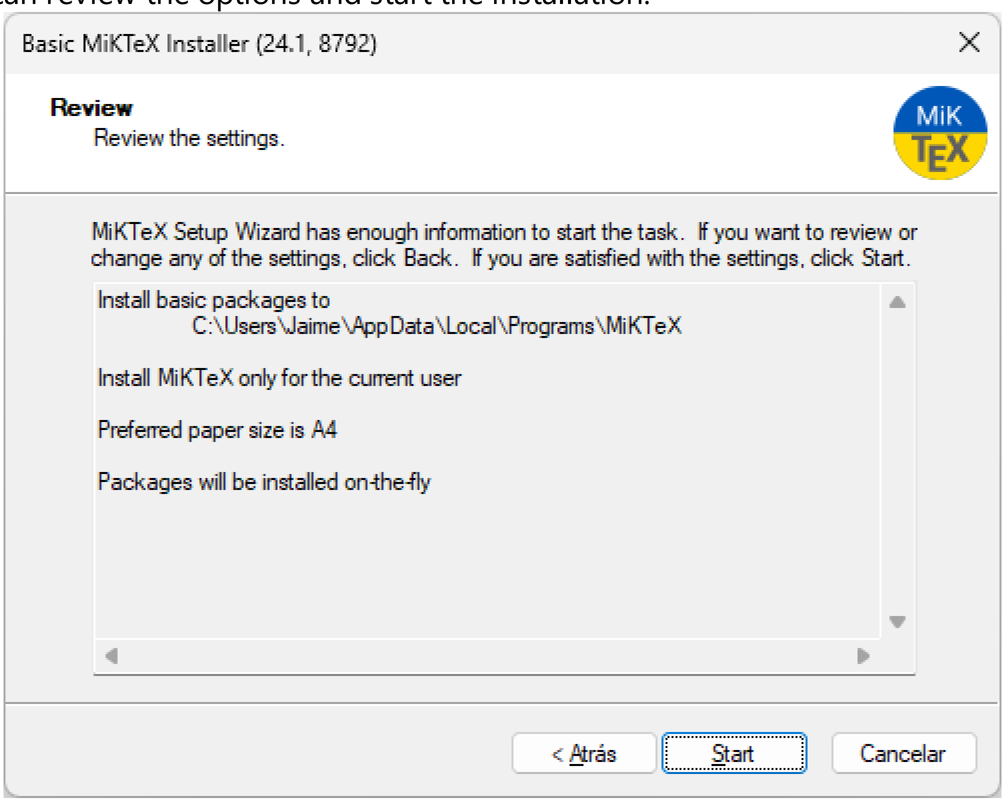
In the next step the user can select where to install it. The decision is up to the user and has no effect on SmartNetics:



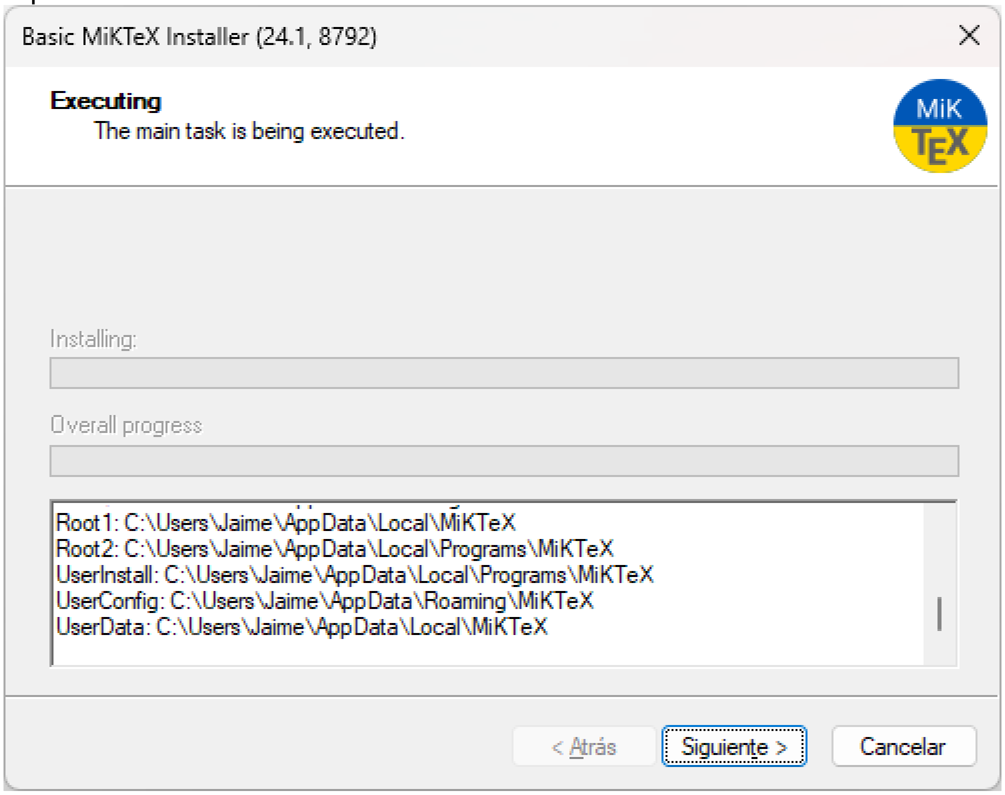
In the next step the user can define the default format and how to install missing packages. The selected format does not affect the output, but **the user is highly encouraged to select <u>"Yes" on "Install missing packages on the fly"</u>, since there are several packages that are used for the PDF creation (this packages are installed only once when the first PDF is created)**:



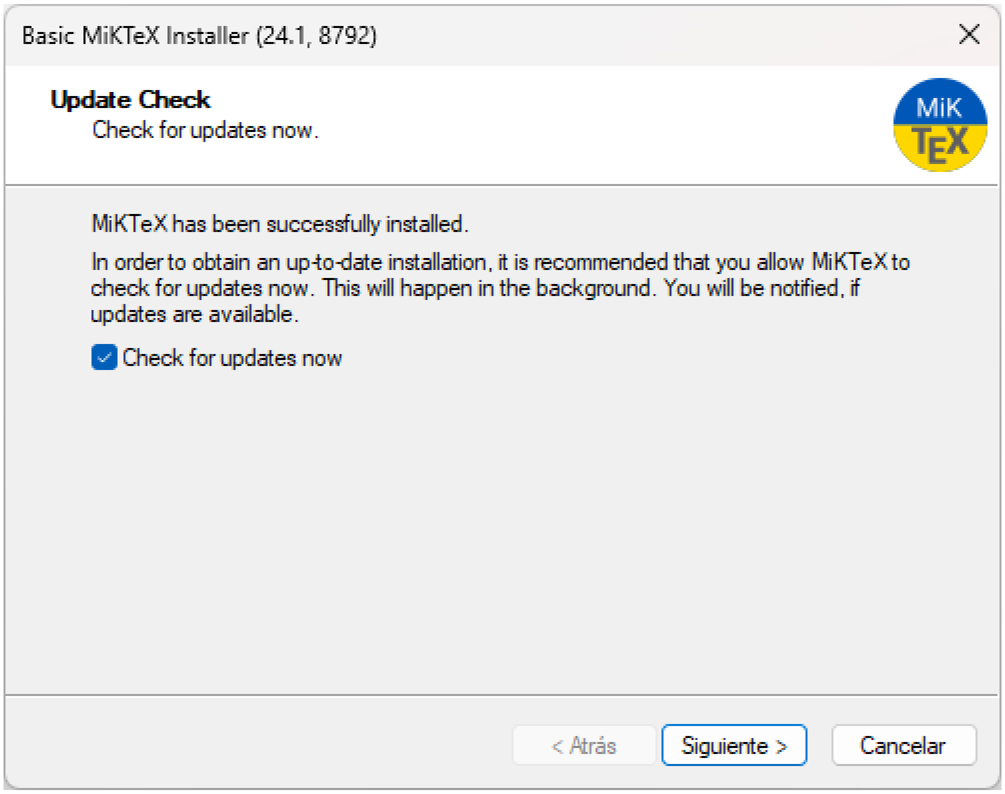
Once the desired configuration is selected, the next screen is displayed, where the user can review the options and start the installation:



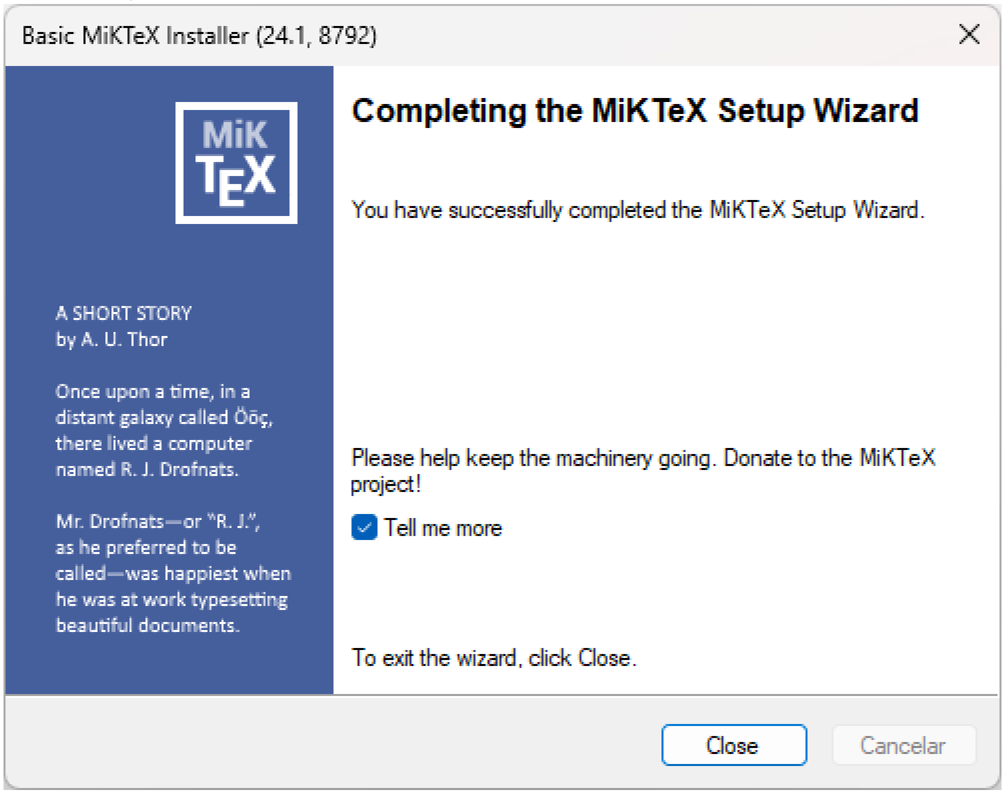
Once the installation finishes, the user can review the results and proceed to the last step:



In the last step the user can select whether or not to check for updates. **the user is highly encouraged to select "Check for updates now" on the first installation**, since it may be required for the compiler to work as expected:



After this, the installation is finished:



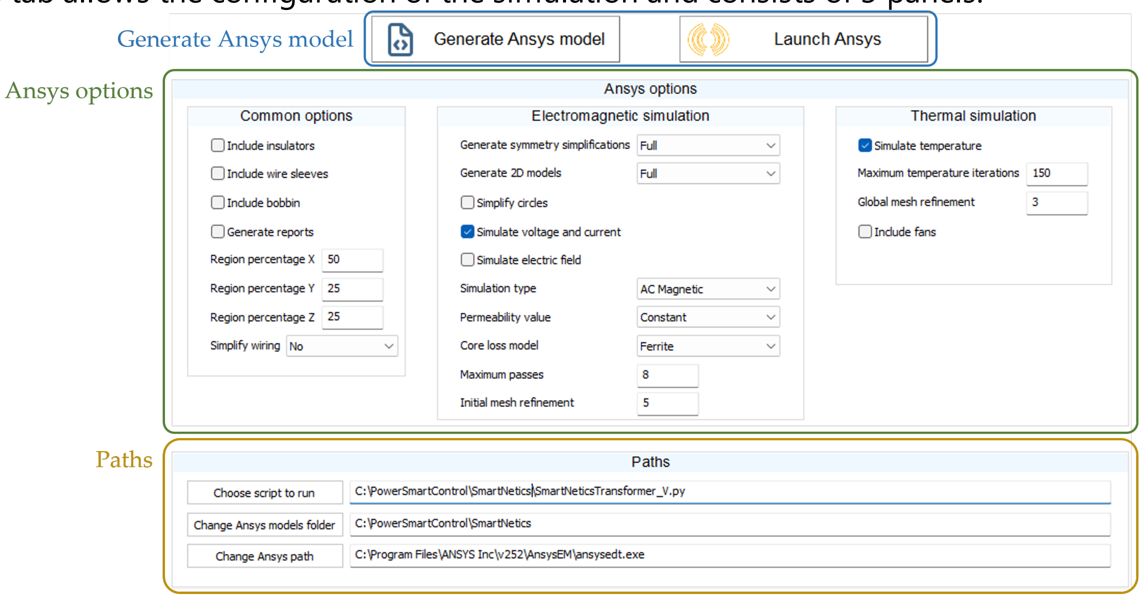
7.5 Ansys

This tab allows the creation of the [Ansys](#) model of the selected device.

SmartNetics allows the direct exportation to Ansys-Maxwell and Ansys-Icepak, where the device can be simulated. To do so, the first step is to generate the model, which is defined in a python script. This is done when clicking on “Generate Ansys model” and no Ansys installation is required.

Once the model is created, the user can launch Ansys themselves and run it, in the same computer SmartNetics is installed or any other with a valid Ansys license. Ansys can also be launched from this same screen, by clicking on “Launch Ansys”. To run the simulation, an installation of Ansys-Maxwell is needed for electromagnetic simulations and of Ansys-Icepak for thermal simulations. Both software products are part of Ansys Electronics Desktop.

This tab allows the configuration of the simulation and consists of 3 panels:



- **Generate Ansys model**: generates the python script that is to be interpreted in Ansys. In this step only the script is generated, so an Ansys installation is not required. The details of the script generation are depicted in the corresponding section.
  Said script can be opened in Ansys (in the machine that generated it or in any other machine with an Ansys installation).
- **Launch Ansys**: launches Ansys and automatically runs the generated script. The script to run can be manually set by the "Choose script to run" button below, once Ansys is launched, the steps to run the simulation and get the results are depicted in

their corresponding section.
*Ansys has to be installed in the machine in the path selected below.

- **Ansys options**: select the options for the building of the model and the simulation configuration:
  - **Common options**
    - **Include insulators**: include the insulators (if any) in the model. Insulators don't have an impact on electromagnetic simulation but they increase meshing and simulation time, so it is advised to only activate this option for electric field or thermal simulations.
    - **Include wire sleeves**: include the wire sleeves in the model. Wire sleeves don't have an impact on electromagnetic simulation but they increase meshing and simulation time, so it is advised to only activate this option for electric field or thermal simulations.
    - **Include bobbin**: include the coil former (if any) in the model. The bobbin doesn't have an impact on electromagnetic simulation but it increases meshing and simulation time, so it is advised to only activate this option for electric field or thermal simulations.
    - **Generate reports**: automatically generate the reports in Ansys along with the simulation model.
    - **Region percentage X**: percentage of the model dimension that the region grows in the X axis.
    - **Region percentage Y**: percentage of the model dimension that the region grows in the Y axis.
    - **Region percentage Z**: percentage of the model dimension that the region grows in the Z axis.
    - **Simplify wiring**: Available options for wiring simplification:
      - **No**: Every wire is independently generated. This options allows a more realistic representation of the device, but produces longer simulations.
      - **Parallels**: The wires in parallel for every turn are generated as a single "equivalent" wire. This option speeds up the simulation, in exchange for the loss of some information, like the accurate current distribution among parallel wires.
        - If this option is selected, SmartNetics does the necessary modifications to the parameters of the material when exporting to Ansys. Equivalent resistivity, thermal conductivity, etc, are calculated, so the parallel equivalent matches the real behavior as closely as possible.
  - **Electromagnetic simulation**
    - **Generate symmetry simplifications**: choose simplifications to be made to the model to reduce simulation time. The available options are depicted in the corresponding topic.

  - **Generate 2D models**: choose if a 2D model is generated and if symmetry simplifications are to be made to said model to reduce simulation time. The available options are depicted in the corresponding topic.
  - **Simplify circles**: if active, instead of accurate round circles, simplified circles, with a finite number of segments, is used. This option usually speeds up the simulation without a noticeable decrease in accuracy.
  - **Simulate voltage and current**: active this check-box to generate an "EddyCurrent" type simulation in Ansys-Maxwell. For inductors current is used as perturbation while for transformers 2 simulations are generated, one with a current perturbation and a second one with a voltage perturbation.
  - **Simulate electric field**: active this check box to generate an "ACConduction" type simulation in Ansys-Maxwell.
  - **Simulation type**: allows the selection of single amplitude, single frequency, sinusoidal waveform ("AC Magnetic" (EddyCurrent in previous versions) type simulation in Ansys-Maxwell) or transient simulation ("Transient" or "ElectricTransient" type simulations in Ansys-Maxwell).
  - **Permeability value**: choose whether to use a constant permeability or the one defined by the materials B-H curve.
  - **Core loss model**: choose whether to use a regular core loss definition (by means of Steinmetz parameters) or to replicate the hysteresis B-H curve.
  - **Maximum passes**: set the maximum number of passes for the Ansys simulation (can also be changed later inside Ansys).
  - **Initial mesh refinement**: choose the initial mesh refinement level. 1 for a very coarse mesh and 9 for a very fine one (can also be changed later inside Ansys).
- **Thermal simulation**
  - **Simulate temperature**: active this check box to generate a thermal simulation in Ansys-Icepak.
  - **Maximum temperature iterations**: set the maximum number of iterations for the temperature calculation (can also be changed later inside Ansys).
  - **Global mesh refinement**: choose the mesh refinement level. 1 for a very coarse mesh and 5 for a very fine one (can also be changed later inside Ansys).
  - **Include fans**: active this check-box to export the fan to Ansys, with the same configuration used for the design. This option only takes effect if a fan was selected in the thermal configuration panel.

- **Paths**: configure the options regarding the paths.
  - **Choose script**: choose script to run if the "Launch Ansys" button is pressed.
  - **Change Ansys models folder**: choose the folder in which the models are going to be generated.
  - **Change Ansys path**: set the patch for the Ansys executable (ansysedt.exe). It is commonly located at "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe".

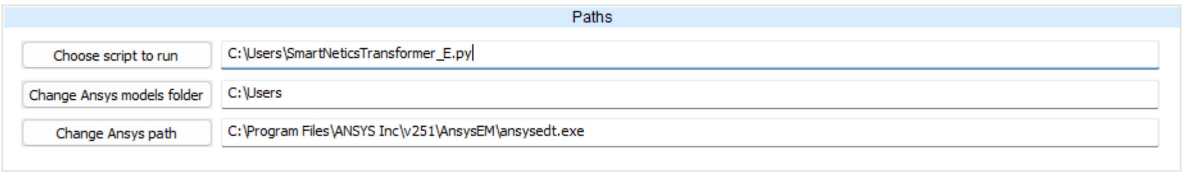
7.5.1 Generate Ansys model

When clicking on 'Generate Ansys model', a number of models is generated.

Depending on the selected options, up to 4 models are generated, each one contained in an independent file:

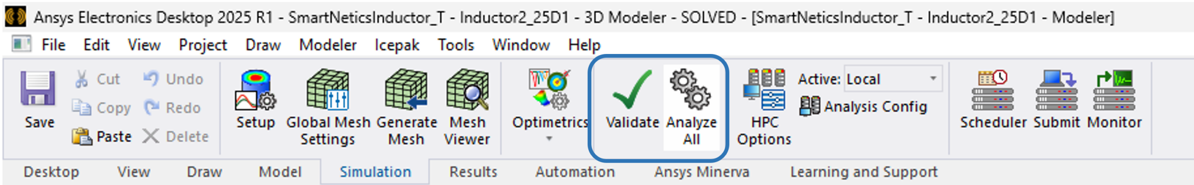
- **SmartNeticsXXXX_I.py:** Maxwell model for EddyCurrent or Transient simulations with current as perturbation.
- **SmartNeticsXXXX_V.py:** Maxwell model for EddyCurrent or Transient simulations with voltage as perturbation (only for transformers).
- **SmartNeticsXXXX_E.py:** Maxwell model for Electric (ACconduction) or ElectricTransient simulations with voltage as perturbation.
- **SmartNeticsXXXX_T.py:** Icepak model for Temperature simulations with power losses as perturbation.

By default, from the ones generated, the first is used as 'Script to run' and its path is shown in the boxes at the bottom.
Nevertheless, the user can select any other model (even one that was generated in a different session) by clicking 'Choose script to run', as shown in the figure:



7.5.2 Run simulation and get results

Once the model is generated, the user can run the simulation and get the desired results.
That is done by clicking the 'Analyze all' button, in the 'Simulation' panel in Ansys, as shown in the next figure.



Some fields are automatically generated when creating the model with SmartNetics, depending on the simulation type:

- **For voltage and current simulations:**
  - B field
  - H field
  - Current density (J)

  - Core loss

- **For electric field simulations:**
  - Voltage

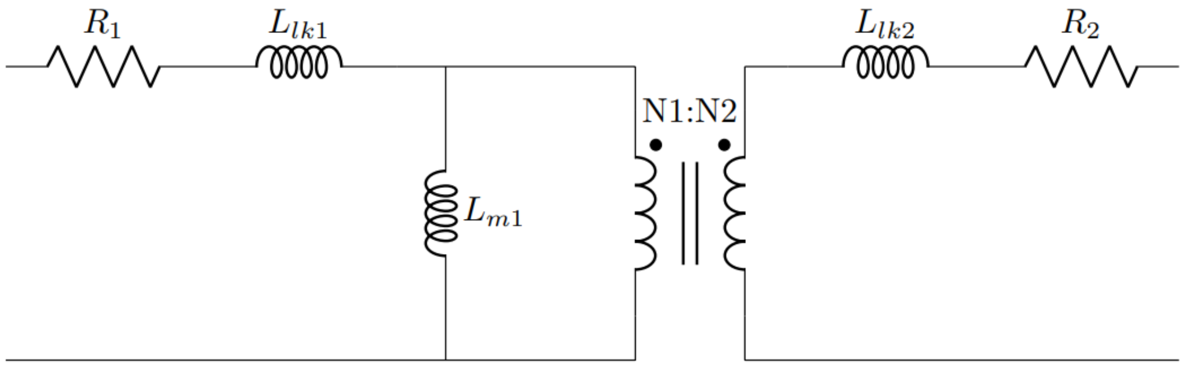
- **For temperature simulations:**
  - Temperature
  - Velocity (only if a fan is included)

If the 'Generate reports' option was selected, some reports are automatically generated as well:

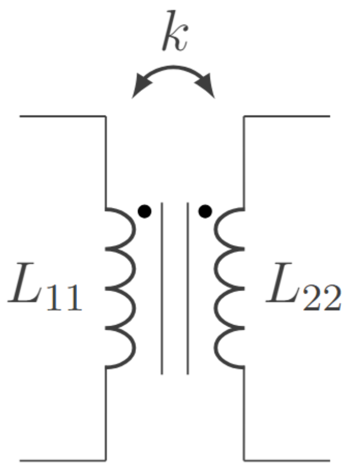
- **For voltage and current simulations:**
  - For transformers:
    - Magnetizing inductance referred to primary (Lm1)
    - Primary leakage inductance (L_lk1)*
    - Primary resistance (R1)*
    - Secondary leakage inductance referred to secondary (L_lk2)*
    - Secondary resistance referred to secondary (R2)*
    - Primary self inductance (L11)
    - Secondary self inductance (L22)
    - Mutual inductance (L12)
    - Coupling coefficient
  - For inductors:
    - Inductance (L)
    - Resistance (R)
  - For both
    - Core loss
    - Winding loss

The equivalent circuit, constructed from the parameters given by the simulation of an inductor is show in the next figure (in the current version, core loss is included in the series R calculation):

The equivalent circuit, constructed from the parameters given by the simulation of a transformer is show in the next figure (in the current version core losses are not included in the transformer model):



A transformer can also be represented as two coupled inductors, defined by their inductance matrix:



$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} L_{11} & L_{12} \\ L_{21} & L_{22} \end{bmatrix} \frac{d}{dt} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix}$$

where the coupling coefficient "k" is defined as:

$$k = \frac{L_{12}}{\sqrt{L_{11} \cdot L_{22}}}$$

*In 2 simulations, on top of the simulated values, a correction is given to increase precision. That correction takes into account the difference between 2D and 3D simulations, allowing fast 2D simulation that provides a result matching the one of a 3D one.

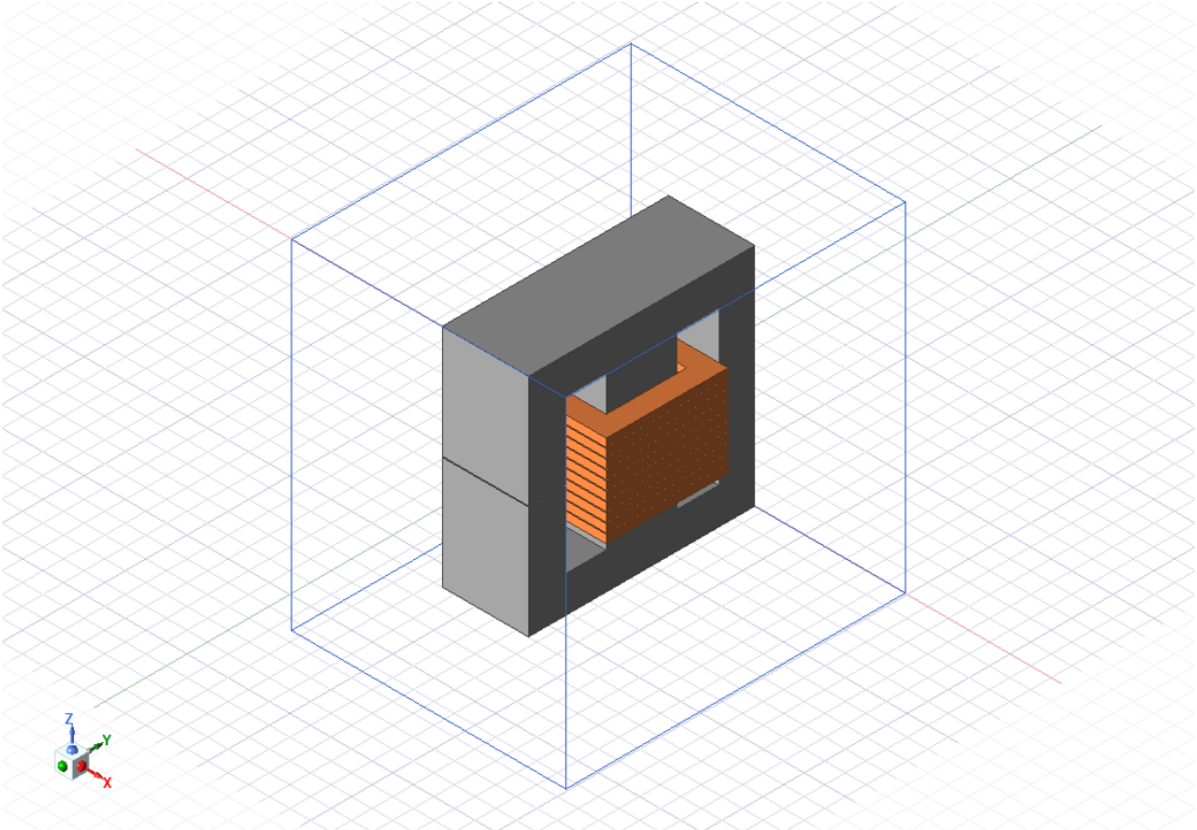
*For voltage and current simulations with 'Transient' simulation type only Core loss and Winding loss are provided

7.5.3 Symmetry simplifications

Thanks to the inherent symmetry of the E core with the winding in the central leg, some simplifications can be made to reduce simulation time while maintaining a good accuracy.

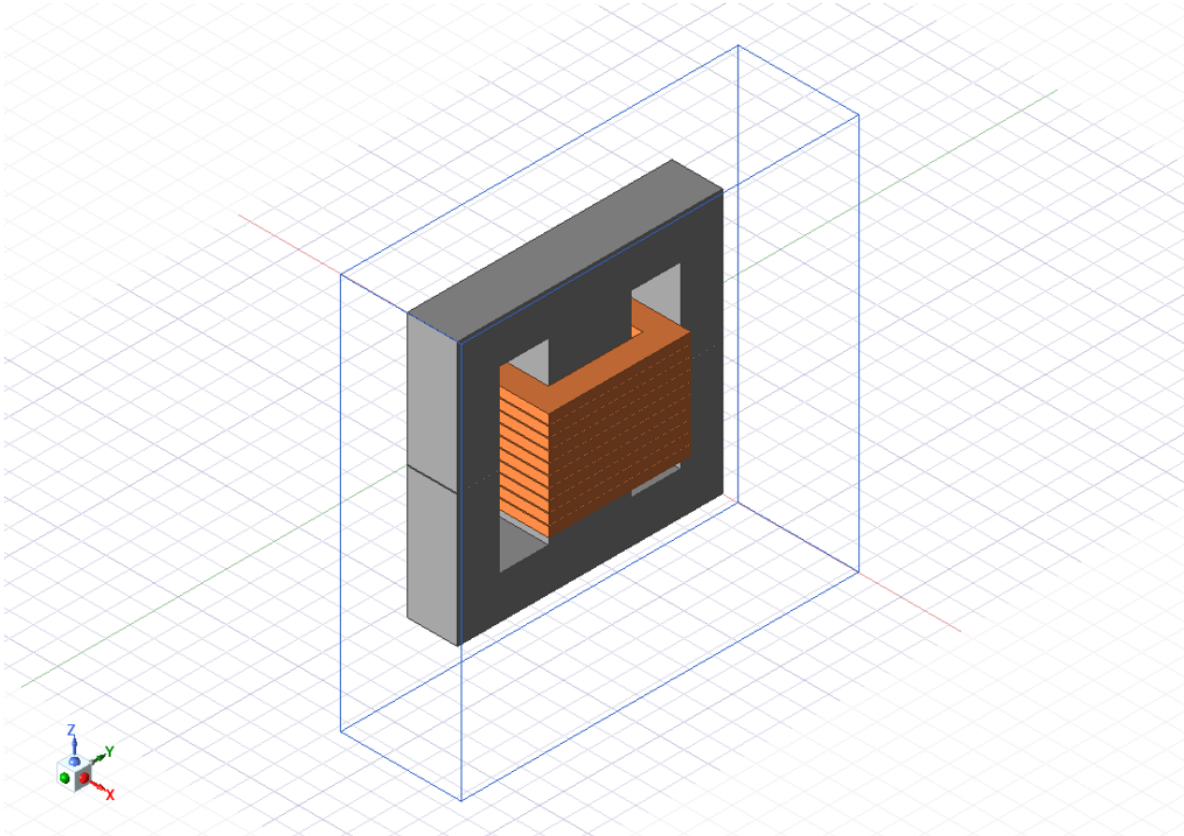
The 3D model is always generated and, from said model, the simplifications are made by slicing the model in 2 using a plane, as depicted in the next images:

3D model



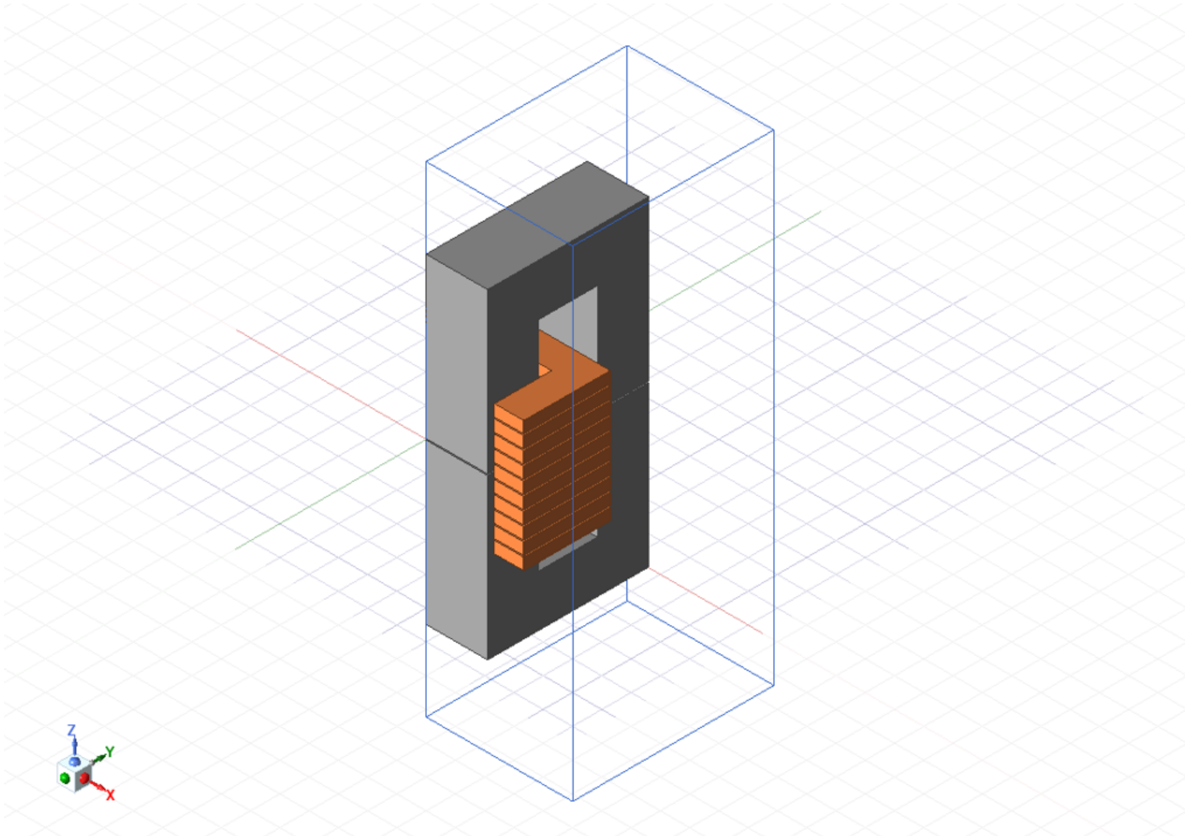
2.5D model (one half of the full model)

Thanks to the front-back symmetry of the 3D model, a new model is created, slicing the 3D one by the YZ plane:



2.25D model (one fourth of the full model)

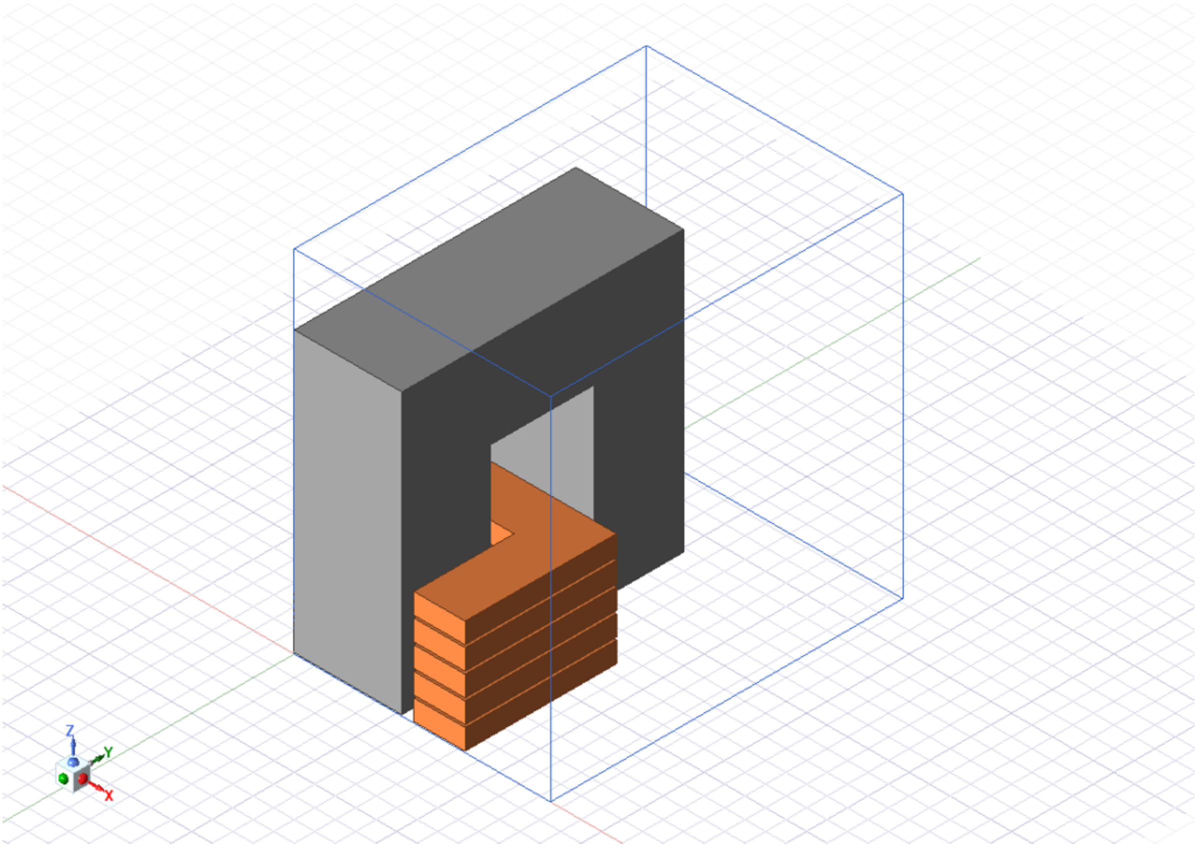
Thanks to the left-right symmetry of the 2.5D model, a new model is created, slicing the 2.5 one by the XZ plane:



2.125D model (one eighth of the full model)

Thanks to the top-bottom symmetry of the 2.25D model, a new model is created, slicing the 2.25 one by the XY plane:

* Take into account that not every model is top-to-bottom symmetric. Only the ones with an even number of turns in every layer keep this symmetry.



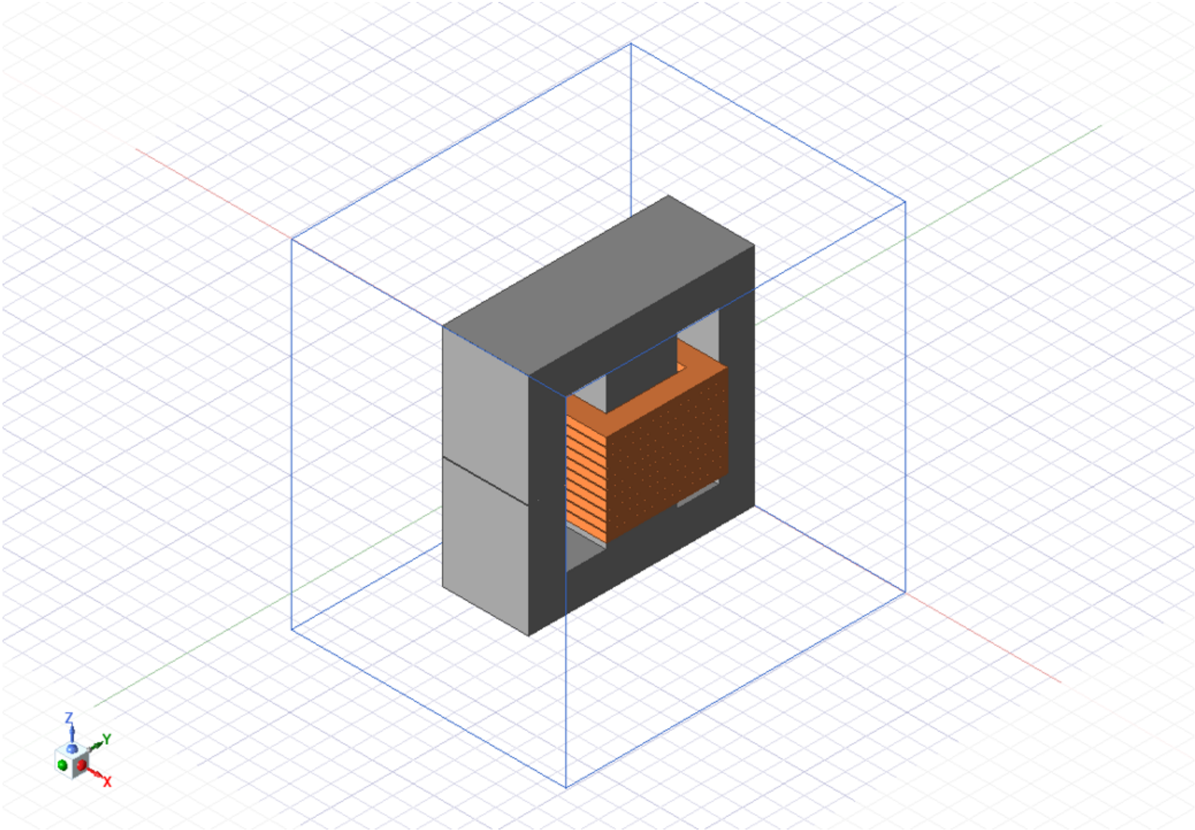
7.5.4 2D models

To reduce simulation time, instead of a full3D model, a 2D slice of it can be used. This simplification reduces the need of time and resources used in the simulation but reduces accuracy.

The 3D model is always generated and, from said model, the 2D is build by slicing it with the YZ plane.

The remaining simplifications come from the 2D model symmetries:

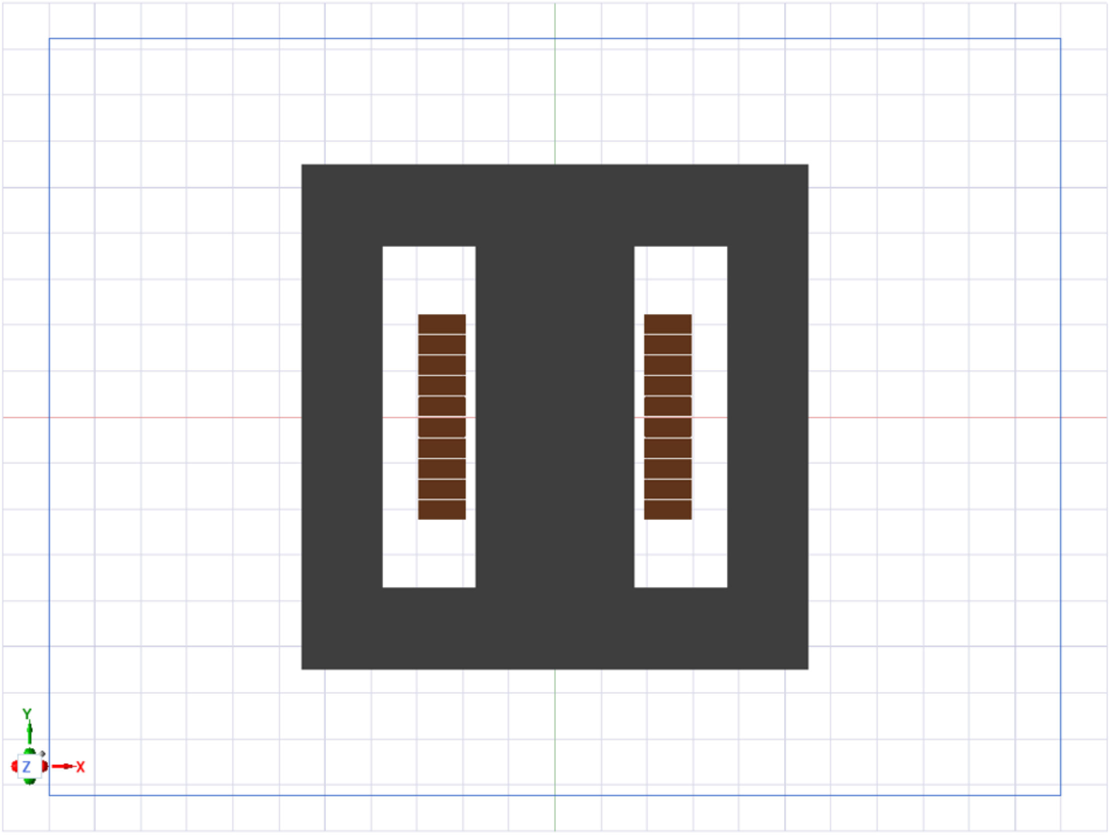
3D model



2D model

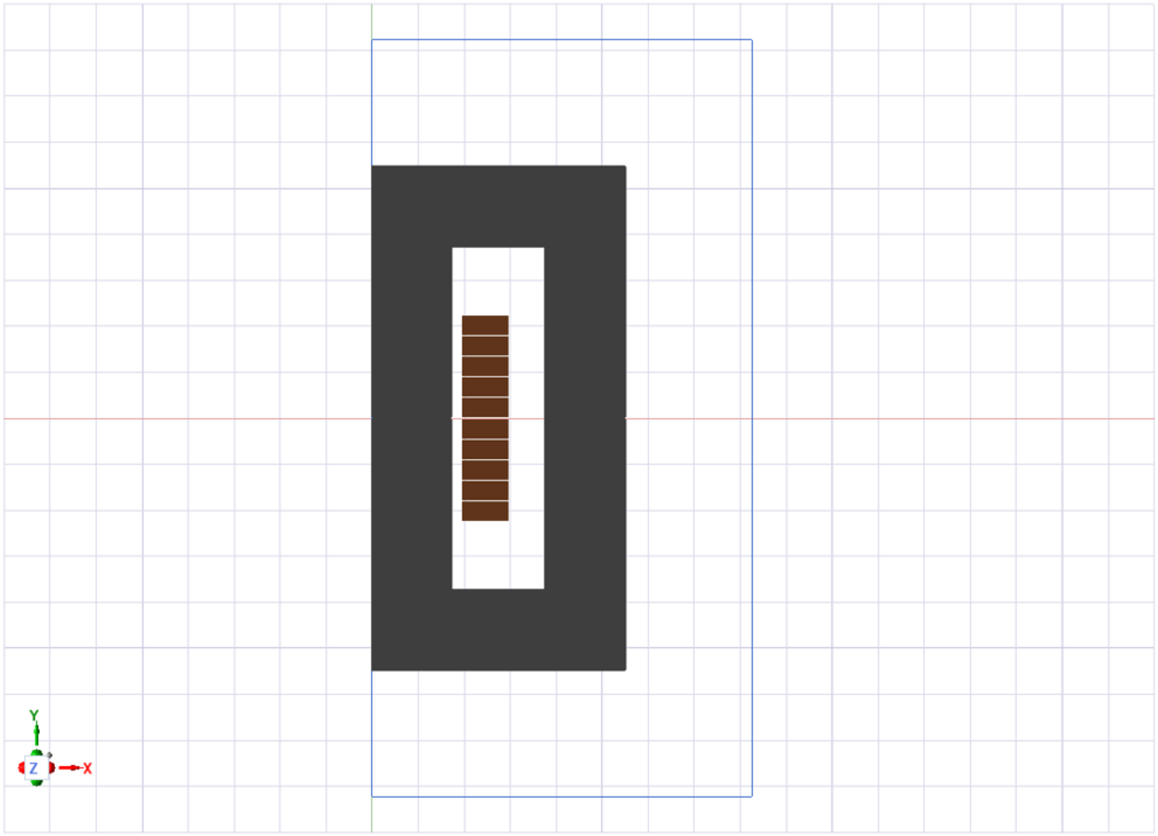
The 2D model is generated by slicing the 3D one by the YZ plane.

*This changes the reference axis from XYZ to XY. The new Y refers to the previous Z and the new X to the previous Y.



1.5D model (1 half of the model)

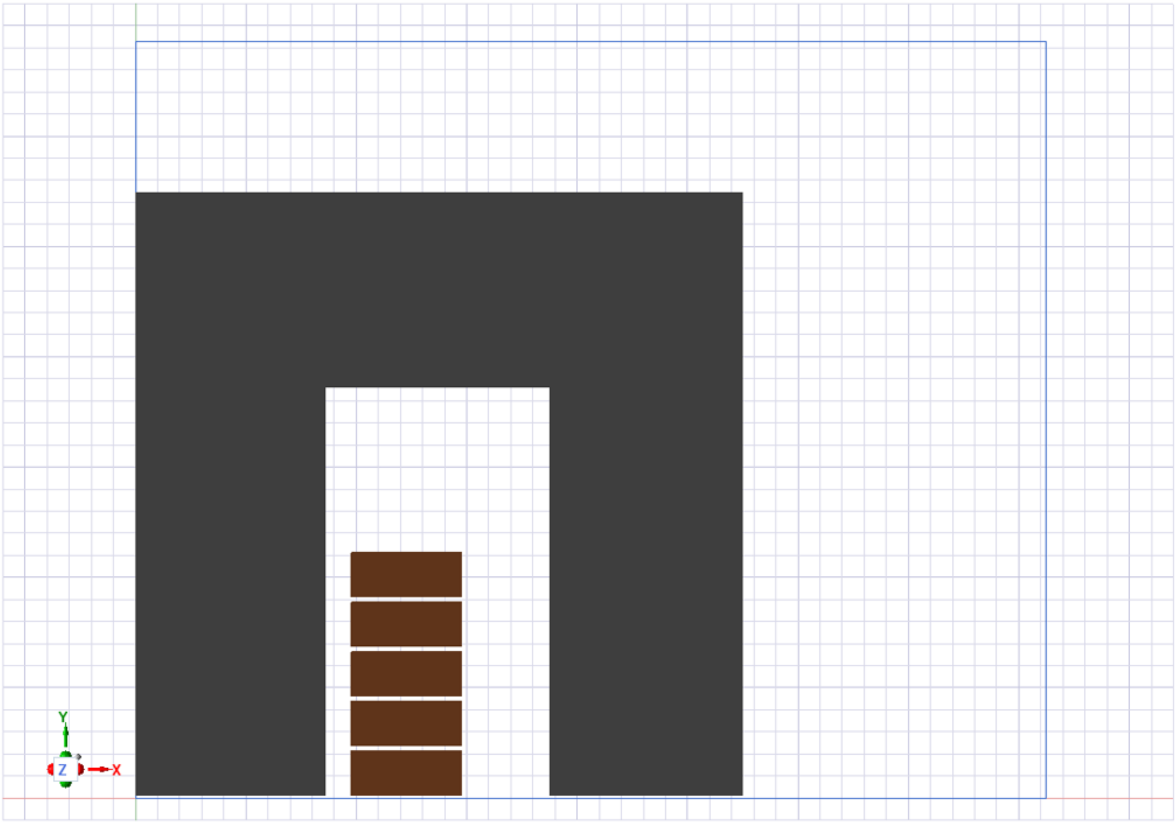
Thanks to the left-right symmetry of the 2D model, a new model is created, slicing the 2 one by the Y axis:



1.25D model

Thanks to the top-bottom symmetry of the 1.5D model, a new model is created, slicing the 1.5 one by the Y axis:

* Take into account that not every model is top-to-bottom symmetric. Only the ones with an even number of turns in every layer keep this symmetry.



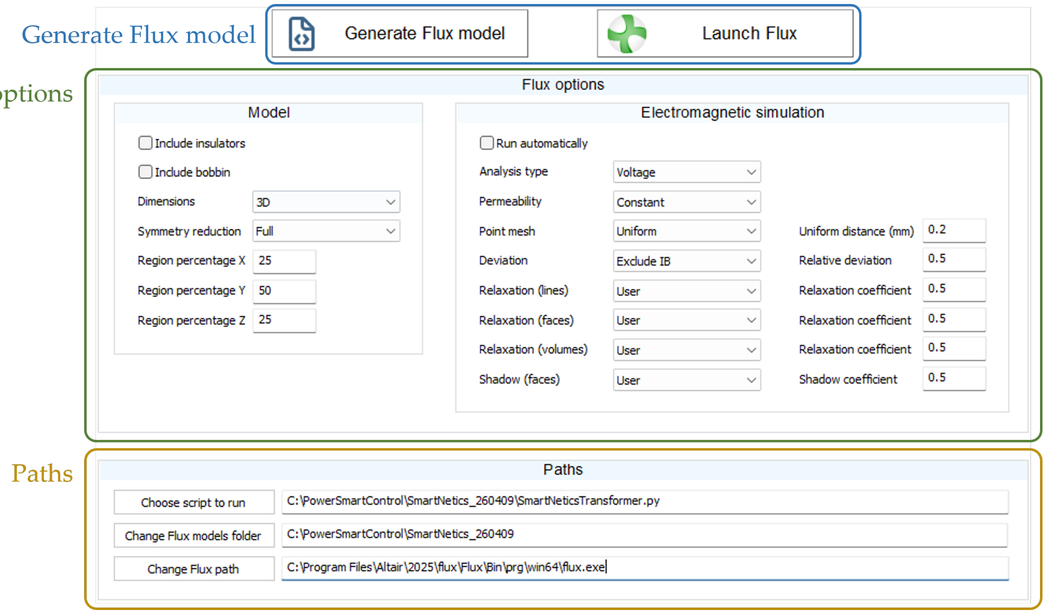
7.6 Altair

This tab allows the creation of the Flux model of the selected device.

SmartNetics allows the direct exportation to Altair-Flux, where the device can be simulated. To do so, the first step is to generate the model, which is defined in a python script. This is done when clicking on “Generate Flux model” and no Flux installation is required.

Once the model is created, the user can launch Flux themselves and run it, in the same computer SmartNetics is installed or any other with a valid Flux license. Flux can also be launched from this same screen, by clicking on “Launch Flux”. To run the simulation, an installation of Altair-Flux is needed.

This tab allows the configuration of the simulation and consists of 3 panels:



- **Generate Flux model**: generates the python script that is to be interpreted in Flux. It can be opened in Flux (in the machine that generated it or in any other machine with a Flux installation).
- **Launch Flux**: launches Flux and automatically runs the generated script. The script to run can be manually set by the "Choose script to run" button below. Flux has to be installed in the machine in the path selected below.

- **Flux options**: select the options for the building of the model and the simulation configuration.:
  - **Model**
    - **Include insulators**: include the insulators (if any) in the model. Insulators don't have an impact on electromagnetic simulation but they increase meshing and simulation time, so it is advised to only activate this option if really needed.
    - **Include bobbin**: include the coil former (if any) in the model. The bobbin doesn't have an impact on electromagnetic simulation but it increases meshing and simulation time, so it is advised to only activate this option if really needed.
    - **Region percentage X**: percentage of the model dimension that the region grows in the X axis.
    - **Region percentage Y**: percentage of the model dimension that the region grows in the Y axis.
    - **Region percentage Z**: percentage of the model dimension that the region grows in the Z axis.
    - **Dimensions**: Select whether to create a 3D or 2D model.
      - Please, notice that, in the current implementation, the 3D model uses non-meshed wires instead of solid one, to reduce simulation time.

        - **Symmetry reduction (only for 3D)**: Select whether to create a full model or use symmetry to generate a half, fourth or eighth model. The available options are depicted in the corresponding topic.
          The 2D simulation currently does not support this symmetry reductions.
    - **Electromagnetic simulation**
        - **Run automatically**: automatically run the simulation upon model creation.
        - **Analysis type**: choose to use current or voltage for the simulation (current is preferred for inductors and voltage for transformers).
        - **Permeability**: choose whether to use a constant permeability or to take saturation into account.
        - **Point mesh**: choose the assisted meshing strategy for free points. Please, see the related topic for more details.
            - **Uniform distance (mm)**: distance between points if Uniform is selected.
        - **Deviation**: choose the assisted meshing strategy corresponding to deviation. Please, see the related topic for more details.
            - **Relative deviation**: Relative deviation if Exclude IB or Include IB are selected.
        - **Relaxation (lines)**: choose the mesh relaxation strategy for free lines. Please, see the related topic for more details.
            - **Relaxation coefficient**: relaxation coefficient for lines if User is selected.
        - **Relaxation (faces)**: choose the mesh relaxation strategy for free faces. Please, see the related topic for more details.
            - **Relaxation coefficient**: relaxation coefficient for faces if User is selected.
        - **Relaxation (volumes)**: choose the mesh relaxation strategy for free volumes. Please, see the related topic for more details.
            - **Relaxation coefficient**: relaxation coefficient for volumes if User is selected.
        - **Shadow (faces)**: choose the shadow strategy for free faces. Please, see the related topic for more details.
            - **Shadow coefficient**: shadow coefficient for volumes if User is selected.

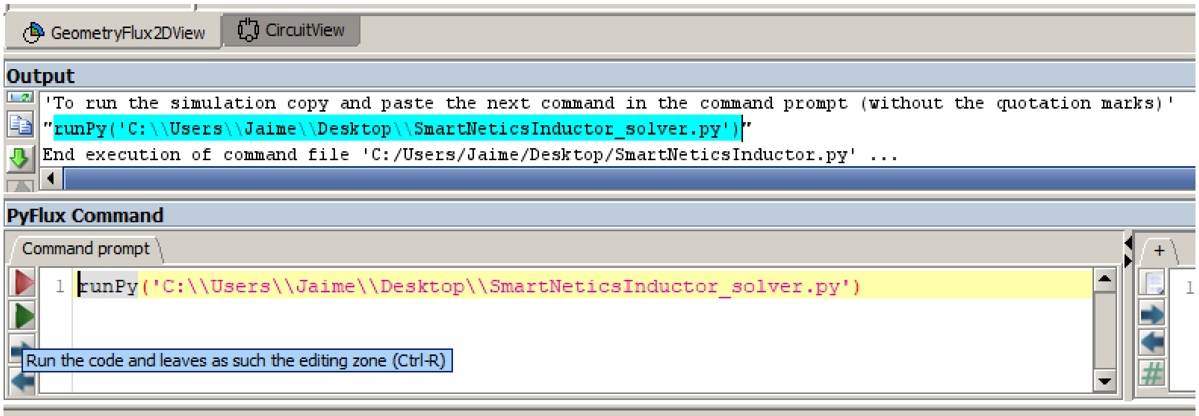
- **Paths**: configure the options regarding the paths.
    - **Choose script**: choose script to run if the "Launch Flux" button is pressed.
    - **Change Flux models folder**: choose the folder in which the models are going to be generated.
    - **Change Flux path**: set the patch for the Flux executable (flux.exe). It is commonly located at "C:\Program Files\Altair\2025\flux\Flux\Bin\prg\win64\flux.exe".

The full Flux documentation can be accessed online, along with its python documentation.

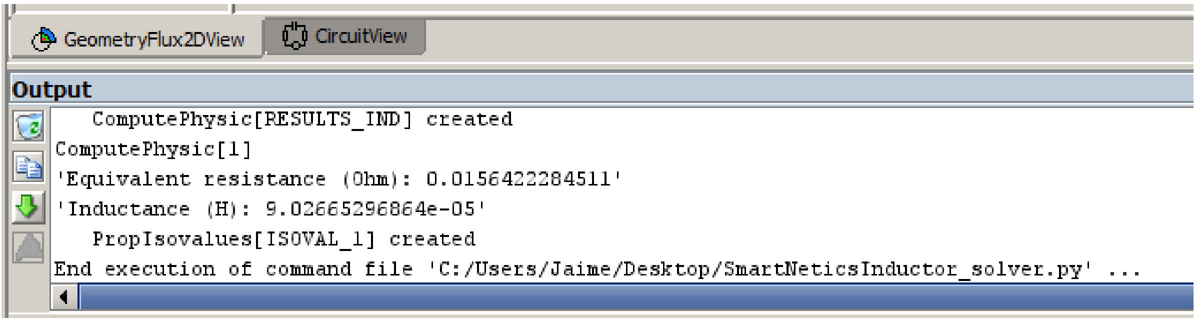
7.6.1 Run simulation and get results

Depending on the state of the 'Run automatically' check-box when pressing 'Generate Flux model', the 'Launch Flux' button has two different behaviors:

- If the 'Run automatically' option is checked, once the user click on 'Launch Flux', Altair-Flux automatically launches, creates the model, runs the simulation and provides the results.
- If the 'Run automatically' option is NOT checked, once the user click on 'Launch Flux', Altair-Flux automatically launches and only creates the model. The user can still run the simulation to get the results by copying the python command written at the "Output" section of Flux, pasting it in its command prompt (without the quotation marks) and pressing the green triangle, as shown below



Once the simulation runs, independently of the selected option, the results are displayed in the 'Output' section of Flux, as shown below:



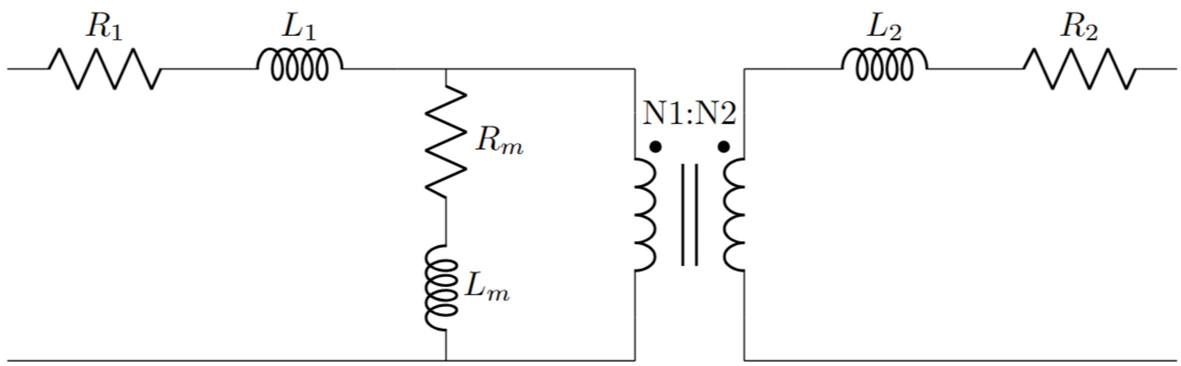
The parameters shown depend on the designed device.
For inductors:

- Inductance
- Equivalent resistance

For transformers:

- Magnetizing inductance (referred to primary side). L_m in the figure below
- Primary leakage inductance. L_1 in the figure below
- Secondary leakage inductance (referred to secondary side). L_2 in the figure below
- Series equivalent magnetizing resistance. R_m in the figure below
- Primary winding equivalent resistance. R_1 in the figure below
- Secondary winding equivalent resistance (referred to secondary side). R_2 in the figure below

The transformer's equivalent circuit is shown below:

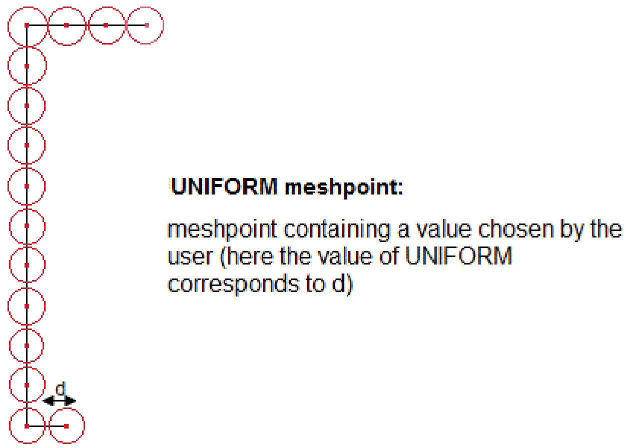


7.6.2 Meshing

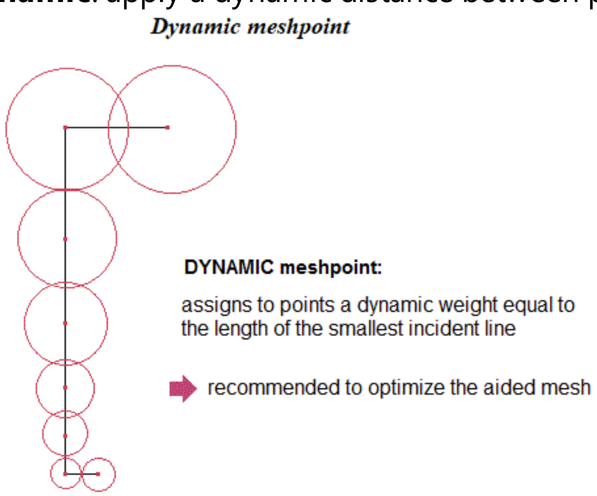
The user can modify many aspects of the mesh to adapt to their particular needs.
The options currently available, along with a brief explanation and images taken from Flux documentation are listed here.

Point mesh: choose the assisted meshing strategy for free points. The available options are:

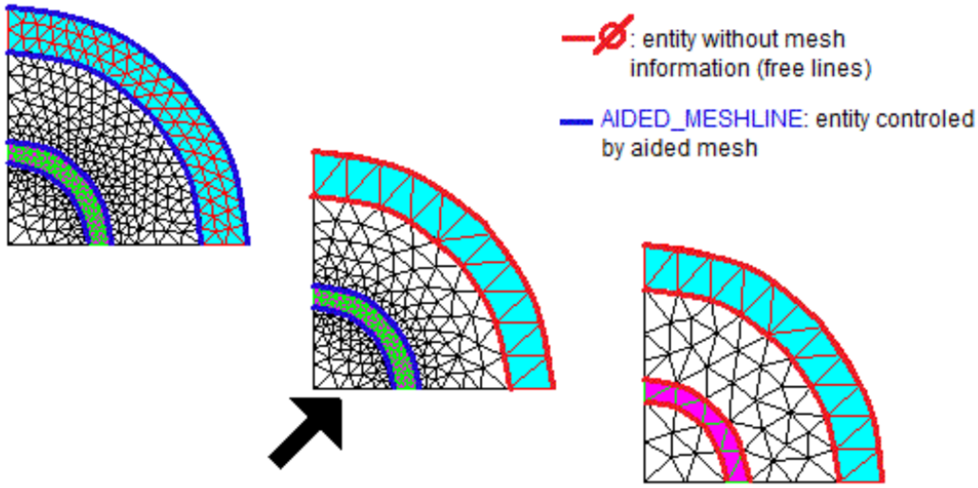
- **Unassign**: no meshing strategy assigned
- **Uniform**: apply a uniform distance between points. From Flux documentation:
  - **Uniform distance (mm)**: uniform distance between points.



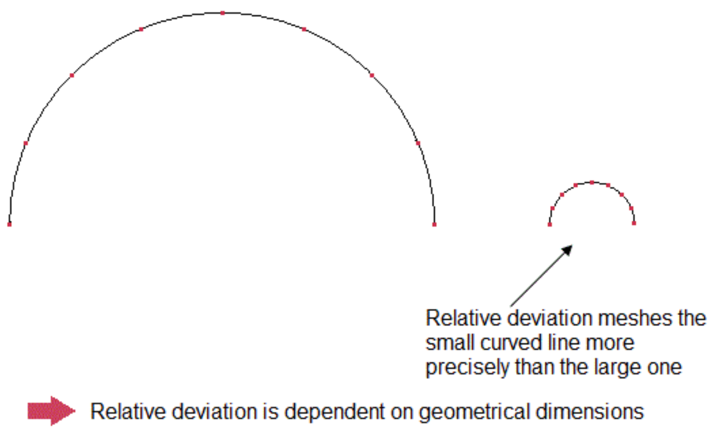
- **Dynamic**: apply a dynamic distance between points. From Flux documentation



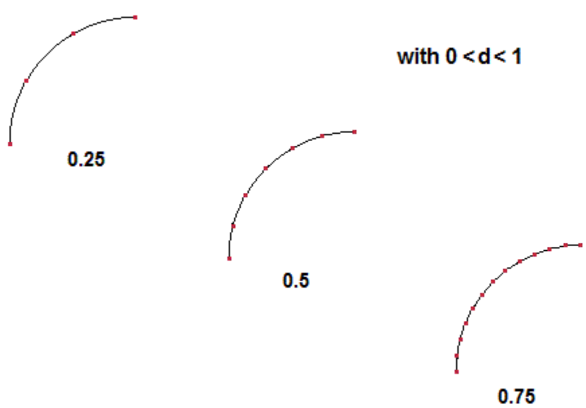
Deviation: choose the assisted meshing strategy corresponding to deviation. From Flux documentation:



Relative deviation



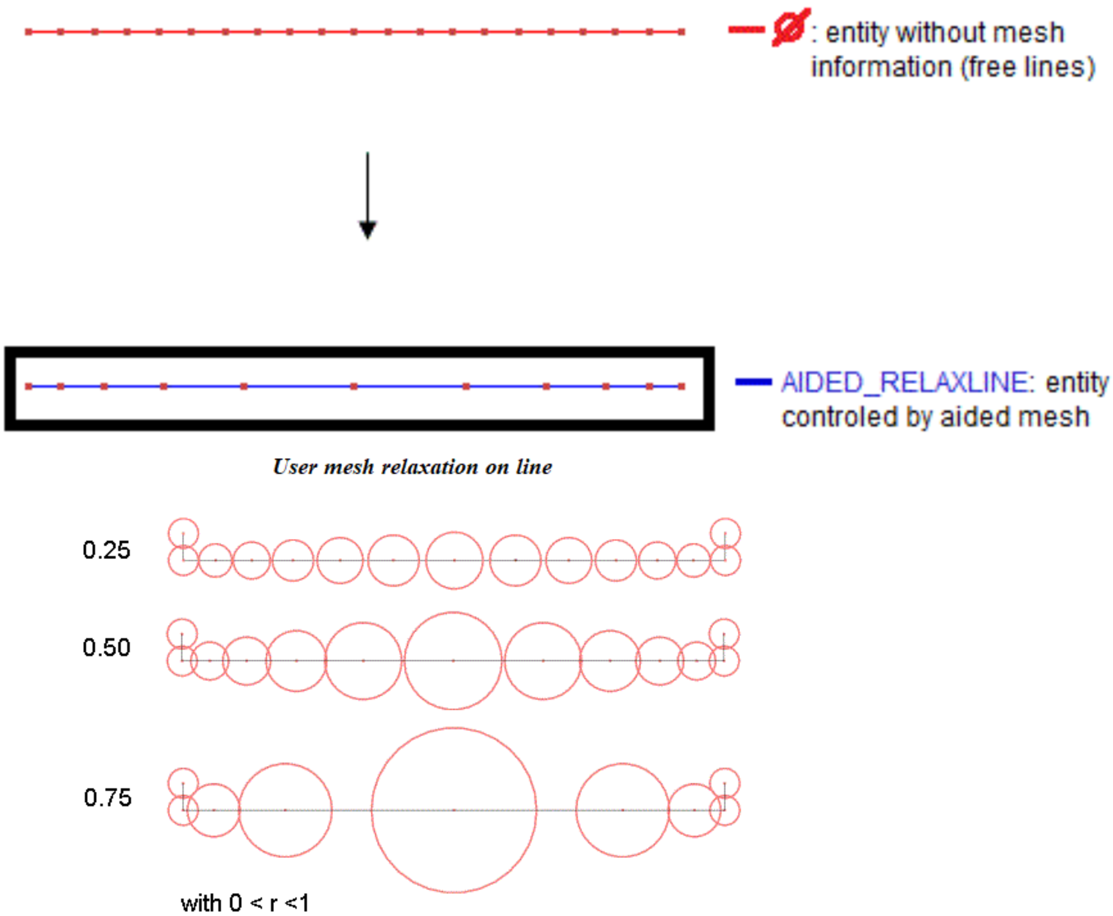
Relative deviation value



The available options are:

- **Unassign**: no deviation strategy assigned.
- **Exclude IB**: exclude the Infinite Box from the deviation setting.
- **Include IB**: include the Infinite Box from the deviation setting.
    - **Relative deviation**: relative deviation if Exclude IB or Include IB are selected.

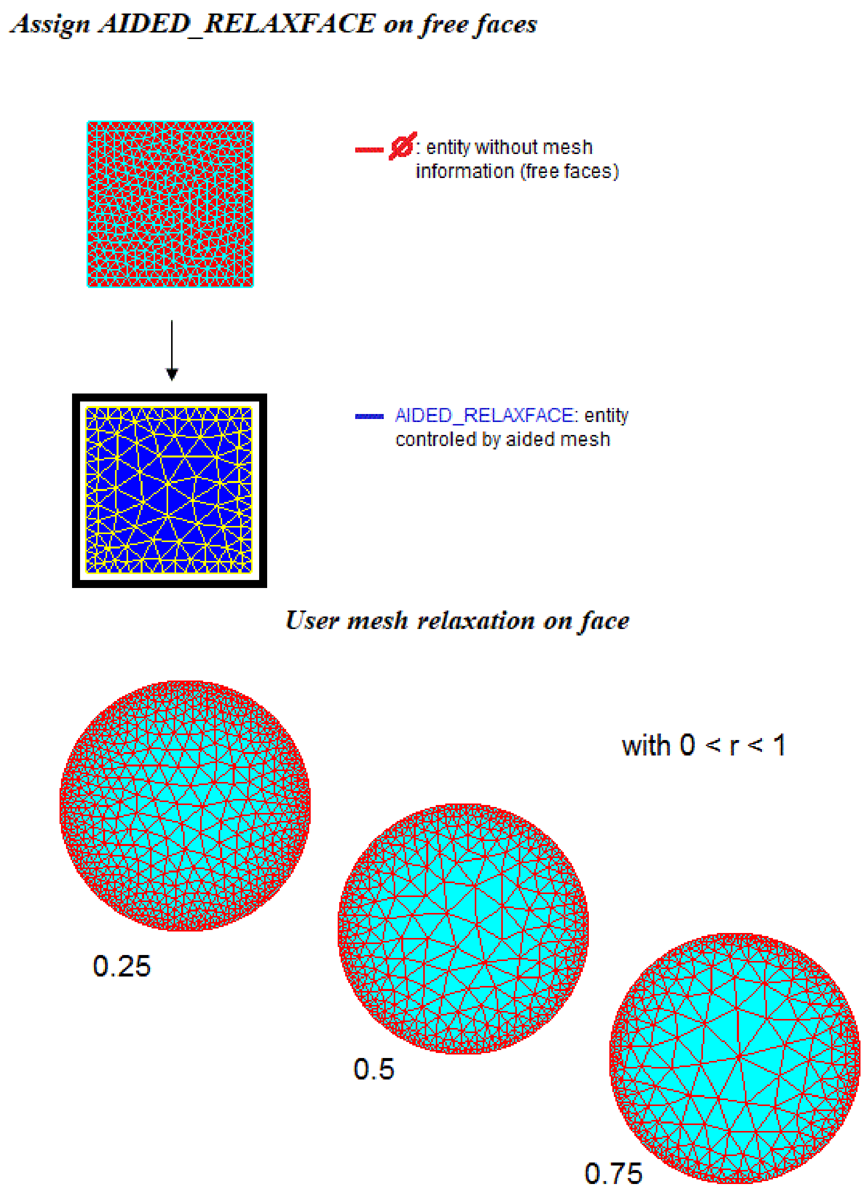
Relaxation (lines): choose the mesh relaxation strategy for free lines. From Flux documentation:



The available options are:

- **Unassign**: no relaxation strategy assigned for free lines.
- **Low**: set a low relaxation coefficient for free lines (equivalent to user relaxation = 0.25).
- **Medium**: set a medium relaxation coefficient for free lines (equivalent to user relaxation = 0.5).
- **High**: set a high relaxation coefficient for free lines (equivalent to user relaxation = 0.75).
- **User**: set a custom relaxation coefficient for free lines.
    - **Relaxation coefficient**: relaxation coefficient for lines if User is selected. A single value from 0 to 1, with a finer mesh the lower the coefficient.

Relaxation (faces): choose the mesh relaxation strategy for free faces. From Flux documentation:



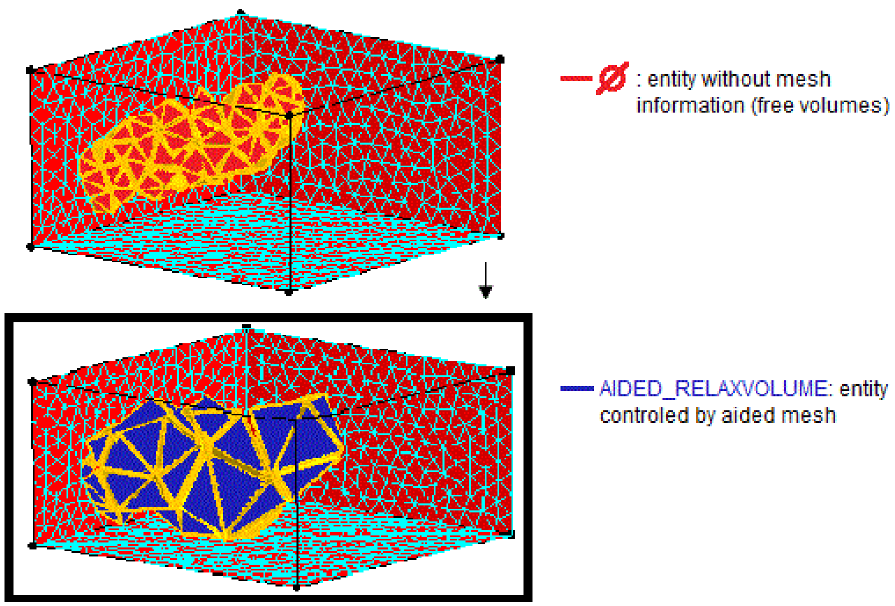
The available options are:

- **Unassign**: no relaxation strategy assigned for free faces.
- **Low**: set a low relaxation coefficient for free faces (equivalent to user relaxation = 0.25).
- **Medium**: set a medium relaxation coefficient for free faces (equivalent to user relaxation = 0.5).
- **High**: set a high relaxation coefficient for free faces (equivalent to user relaxation = 0.75).
- **User**: set a custom relaxation coefficient for free faces.

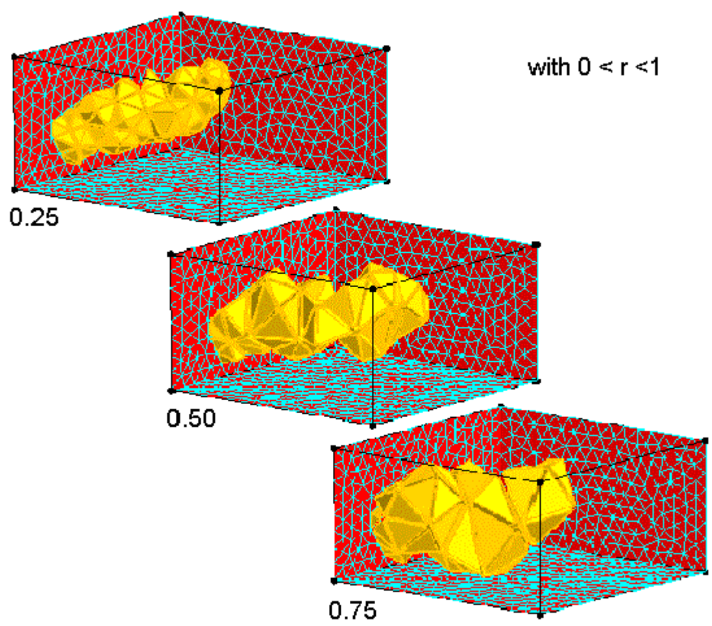
- **Relaxation coefficient**: relaxation coefficient for faces if User is selected. A single value from 0 to 1, with a finer mesh the lower the coefficient.

Relaxation (volumes) (only in 3D): choose the mesh relaxation strategy for free volumes. From Flux documentation:

Assign AIDED_RELAXVOLUME on free volumes



User mesh relaxation on volume



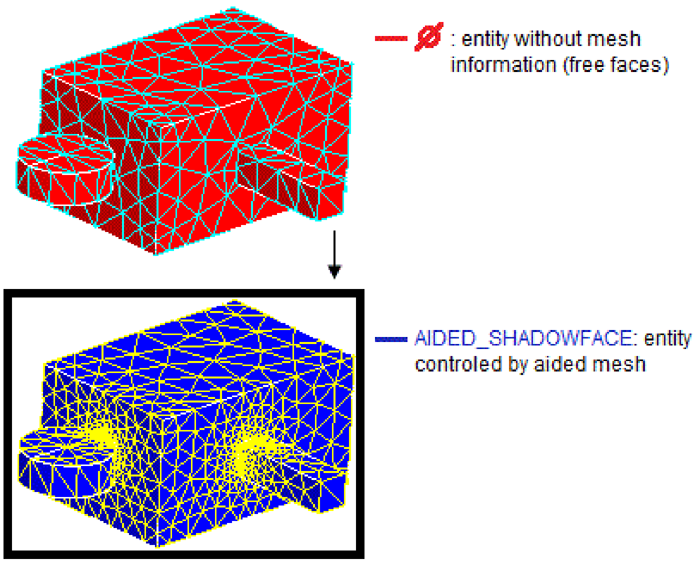
The available options are:

- **Unassign**: no relaxation strategy assigned for free volumes.

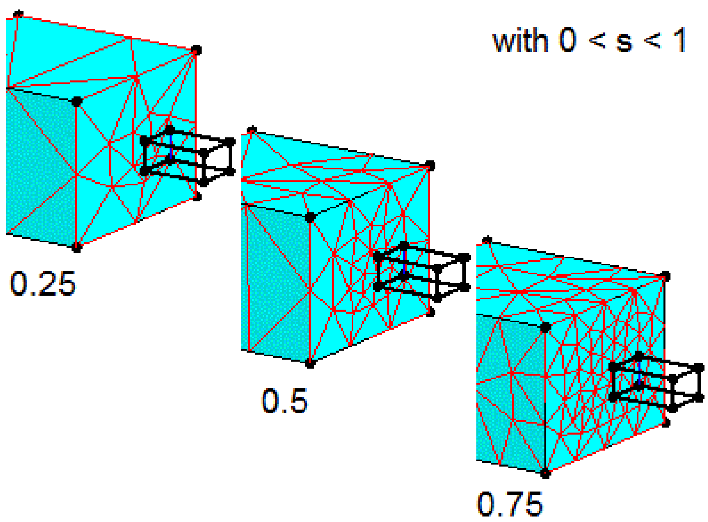
- **Low**: set a low relaxation coefficient for free volumes (equivalent to user relaxation = 0.25).
- **Medium**: set a medium relaxation coefficient for free volumes (equivalent to user relaxation = 0.5).
- **High**: set a high relaxation coefficient for free volumes (equivalent to user relaxation = 0.75).
- **User**: set a custom relaxation coefficient for free volumes.
    - **Relaxation coefficient**: relaxation coefficient for volumes if User is selected. A single value from 0 to 1, with a finer mesh the lower the coefficient.

Shadow (faces) (only in 3D): choose the mesh shadow strategy for free faces. From Flux documentation:

Assign AIDED_SHADOWFACE on free faces



User mesh shadow



The available options are:

- **Unassign**: no shadow strategy assigned for free faces.
- **Low**: set a low shadow coefficient for free faces (equivalent to user relaxation = 0.25).
- **Medium**: set a shadow relaxation coefficient for free faces (equivalent to user relaxation = 0.5).
- **High**: set a high shadow coefficient for free faces (equivalent to user relaxation = 0.75).

- **User**: set a custom shadow coefficient for free faces.
    - **Shadow coefficient**: shadow coefficient for faces if User is selected. A single value from 0 to 1, with a thicker mesh the lower the coefficient.

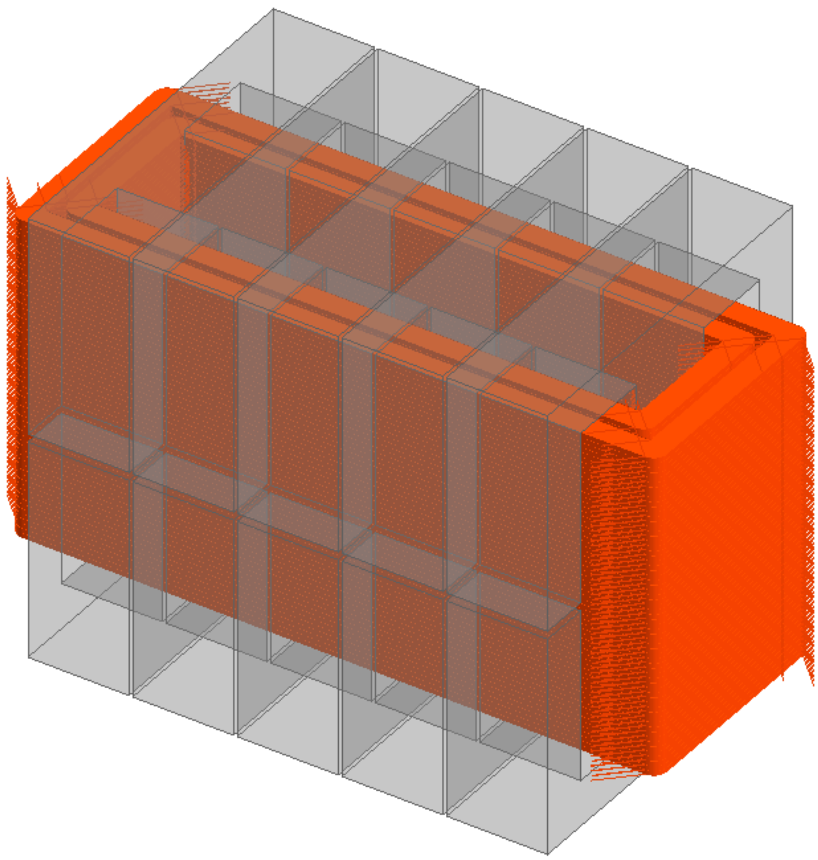
7.6.3 Symmetry simplifications

Thanks to the inherent symmetry of the E core with the winding in the central leg, some simplifications can be made to reduce simulation time while maintaining a good accuracy.

Only the selected symmetry simplification is generated, being all of them one half of the previous one:

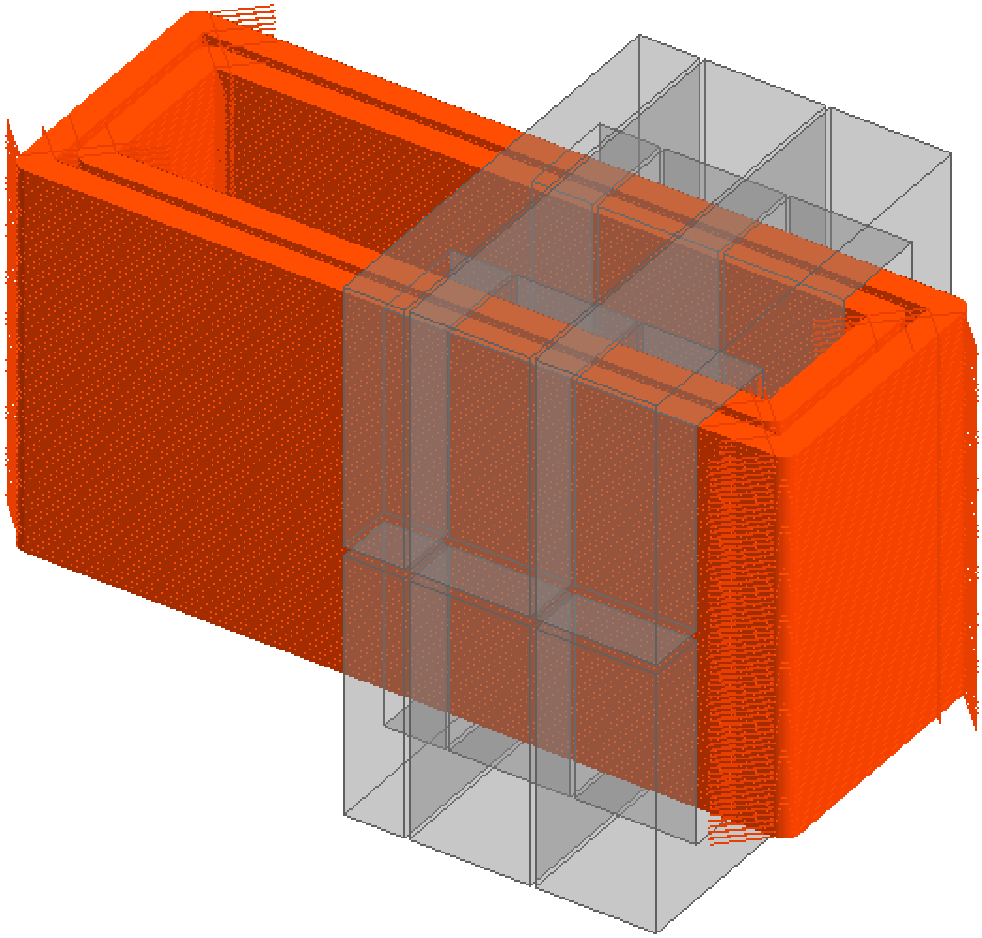
*In the current implementation, for any 3D simulation the windings are defined as non-meshed coils to reduce simulation time

Full model



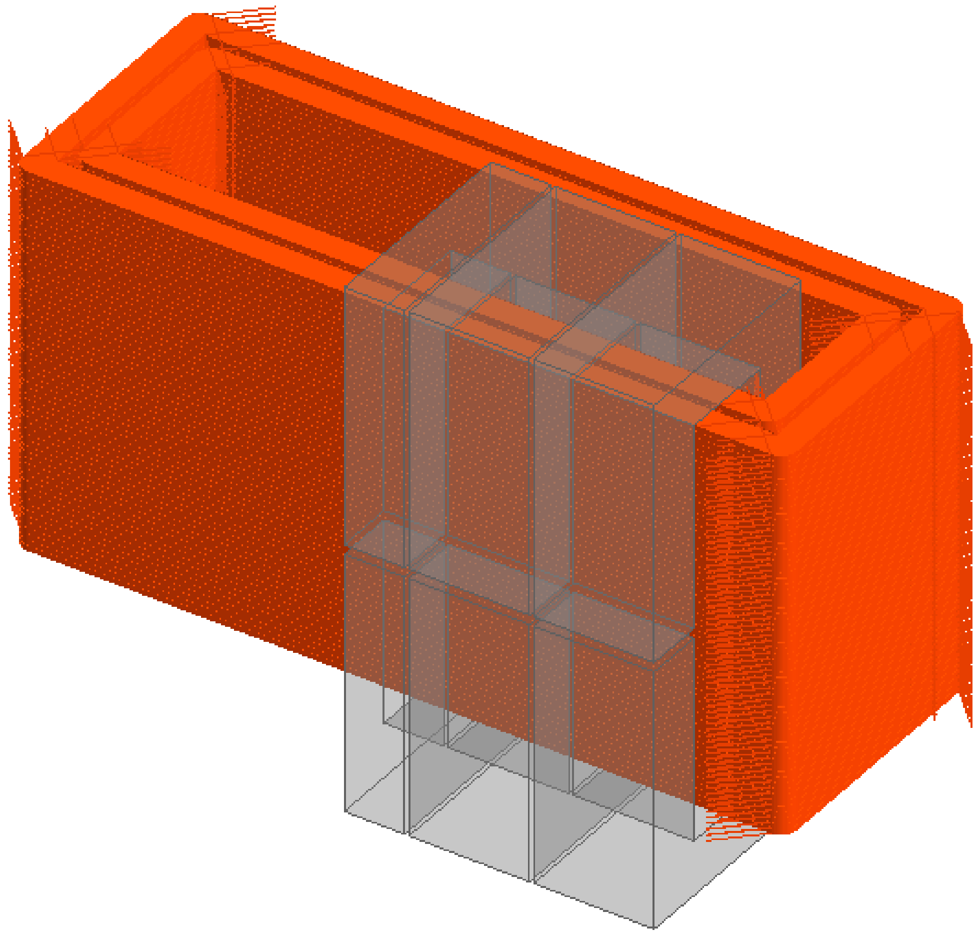
One half of the full model

Thanks to the front-back symmetry of the 3D model, a new model is created, slicing the 3D one by the YZ plane:



One fourth of the full model

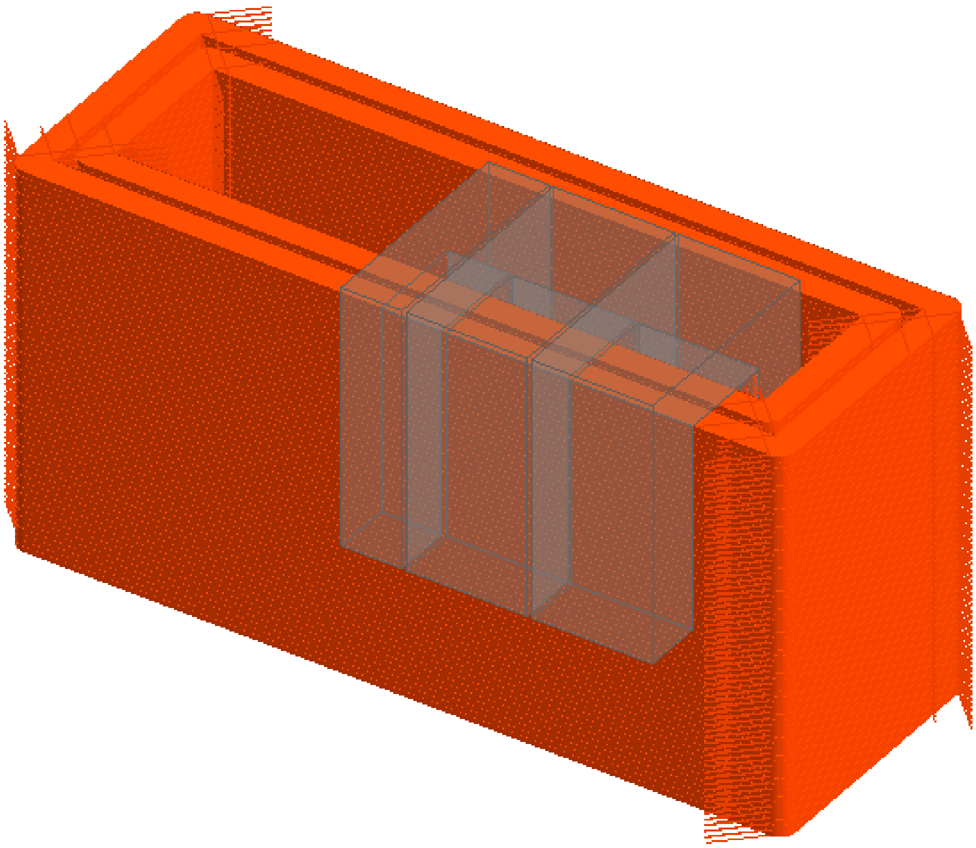
Thanks to the left-right symmetry of the 2.5D model, a new model is created, slicing the 2.5 one by the XZ plane:



One eighth of the full model

Thanks to the top-bottom symmetry of the 2.25D model, a new model is created, slicing the 2.25 one by the XY plane:

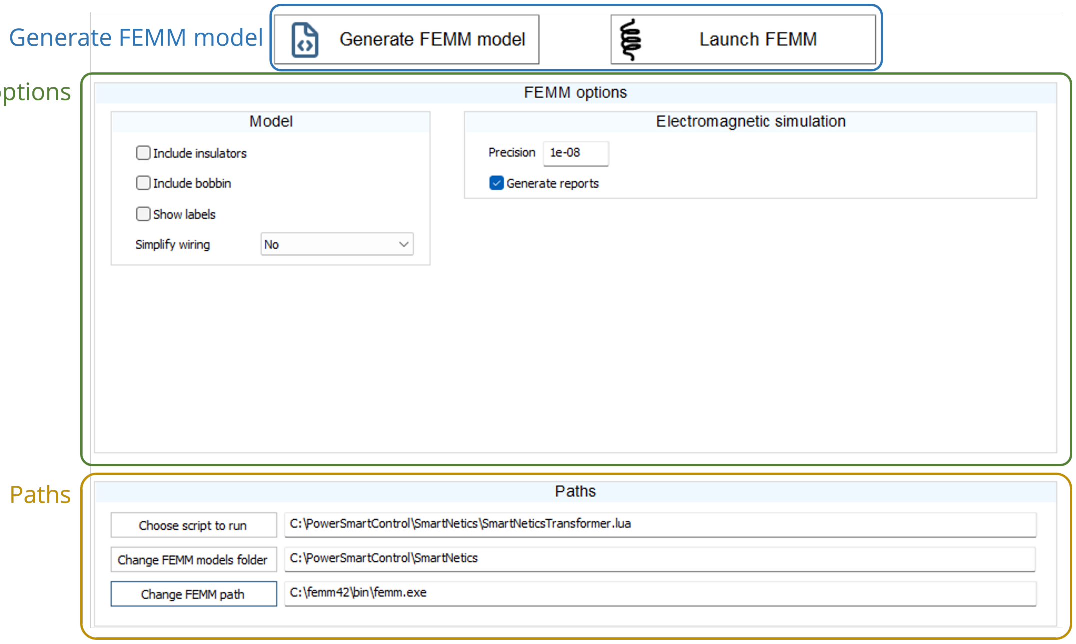
* Take into account that not every model is top-to-bottom symmetric. Only the ones with an even number of turns in every layer keep this symmetry.



7.7 FEMM

This tab allows the creation of the FEMM model of the selected device.

It consists of 3 panels:



- **Generate FEMM model**: generates the lua script that is to be interpreted in FEMM. It can be opened in FEMM (in the machine that generated it or in any other machine with a FEMM installation).
- **Launch FEMM**: launches FEMM and automatically runs the generated script. The script to run can be manually set by the "Choose script to run" button below. FEMM has to be installed in the machine in the path selected below.

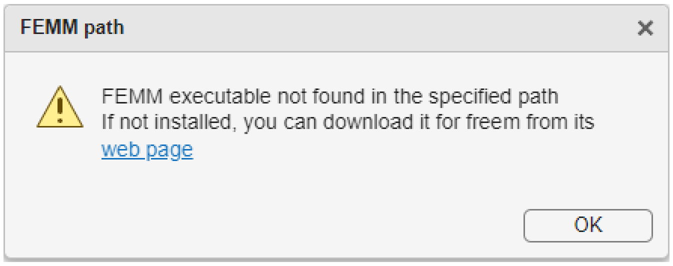
- **FEMM options**: select the options for the building of the model and the simulation configuration.:
  - **Common options**
    - **Include insulators**: include the insulators (if any) in the model.
    - **Include bobbin**: include the coil former (if any) in the model.
    - **Show labels**: show the label assigned to every block created.
  - **Electromagnetic simulation**
    - **Precision**: precision required for the simulation.
    - **Generate reports**: automatically generate the reports in Ansys along with the simulation model.

- **Paths**: configure the options regarding the paths.
  - **Choose script**: choose script to run if the "Launch FEMM" button is pressed.
  - **Change FEMM models folder**: choose the folder in which the models are going to be generated.
  - **Change FEMM path**: set the patch for the Ansys executable (femm.exe).

7.7.1 Installation

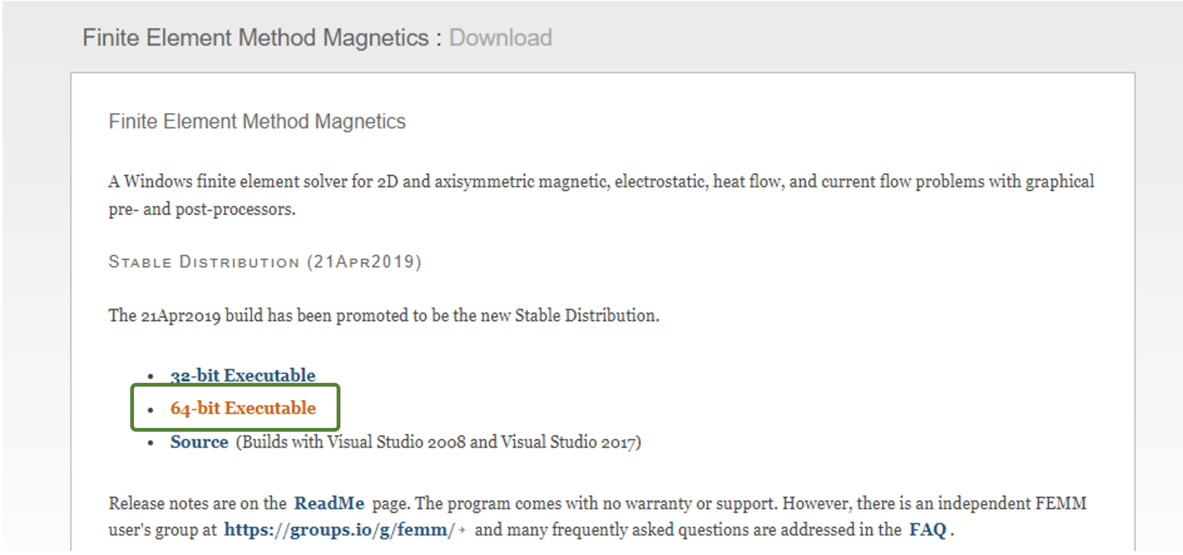
FEMM can be downloaded for free from its web page.

In case no FEMM installation is found in the specified path, a message will be displayed showing the link for its free download:

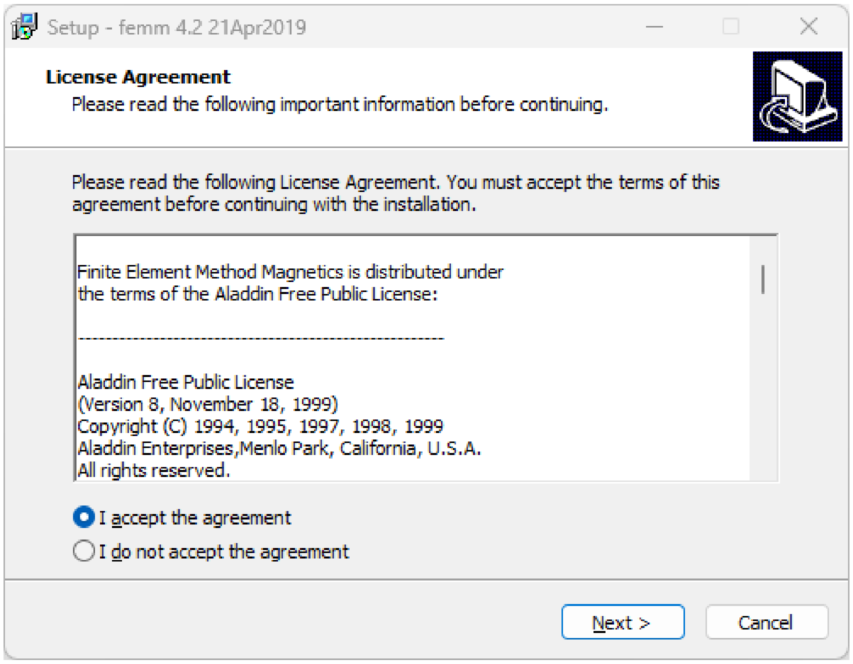


Upon clicking the link, the FEMM download web will open and the next steps can be followed for its installation.

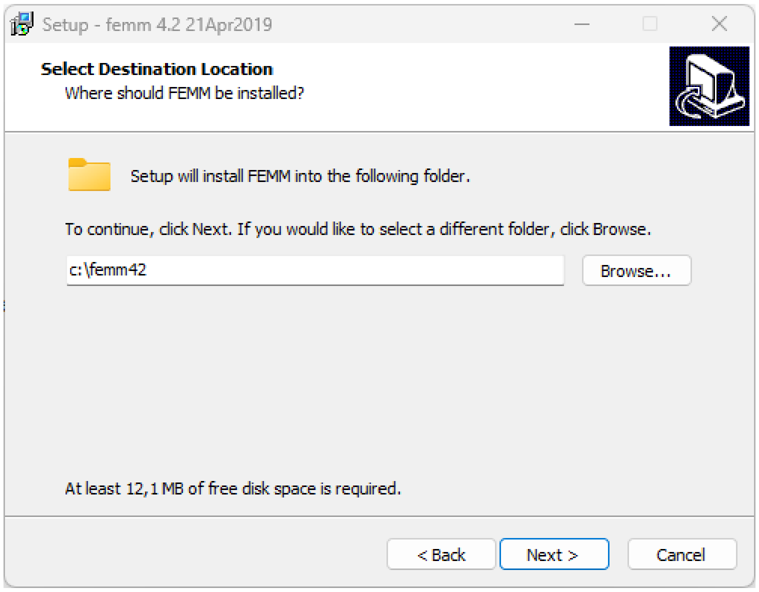
The executable can be downloaded for your current Windows installation (64-bit is the most common):



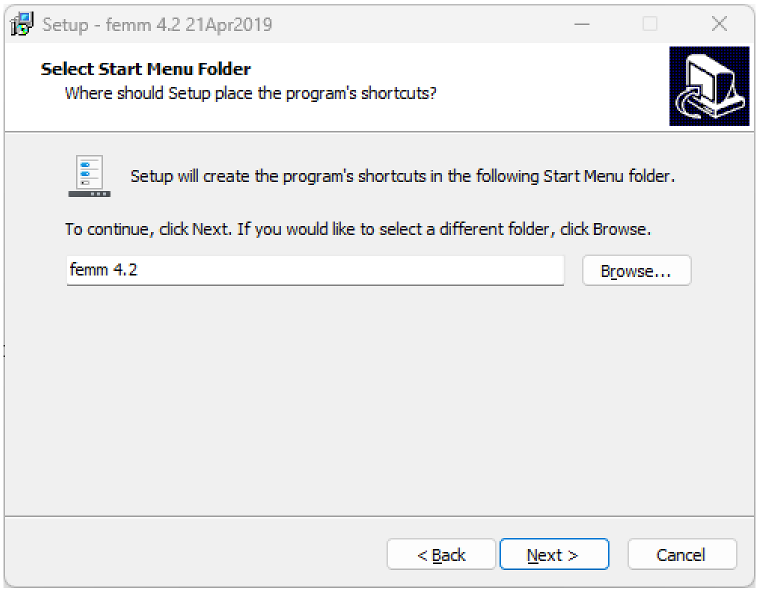
Once the installer is launched, the user needs to accept the License Agreement to carry on the installation:



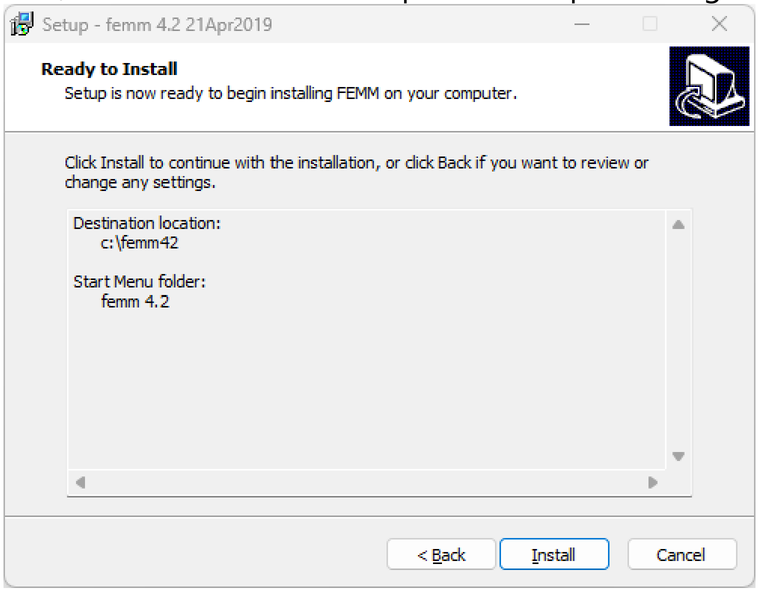
If the user accepts, they can select where to install it. The decision is up to the user and has no effect on SmartNetics:



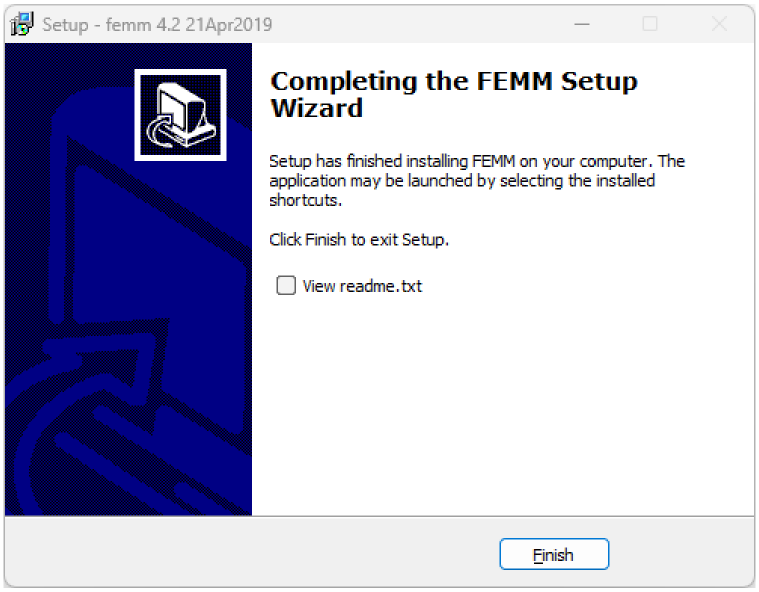
After that, the name for the Start menu shortcut can be selected:



Last, the user can review the previous steps and begin the installation:



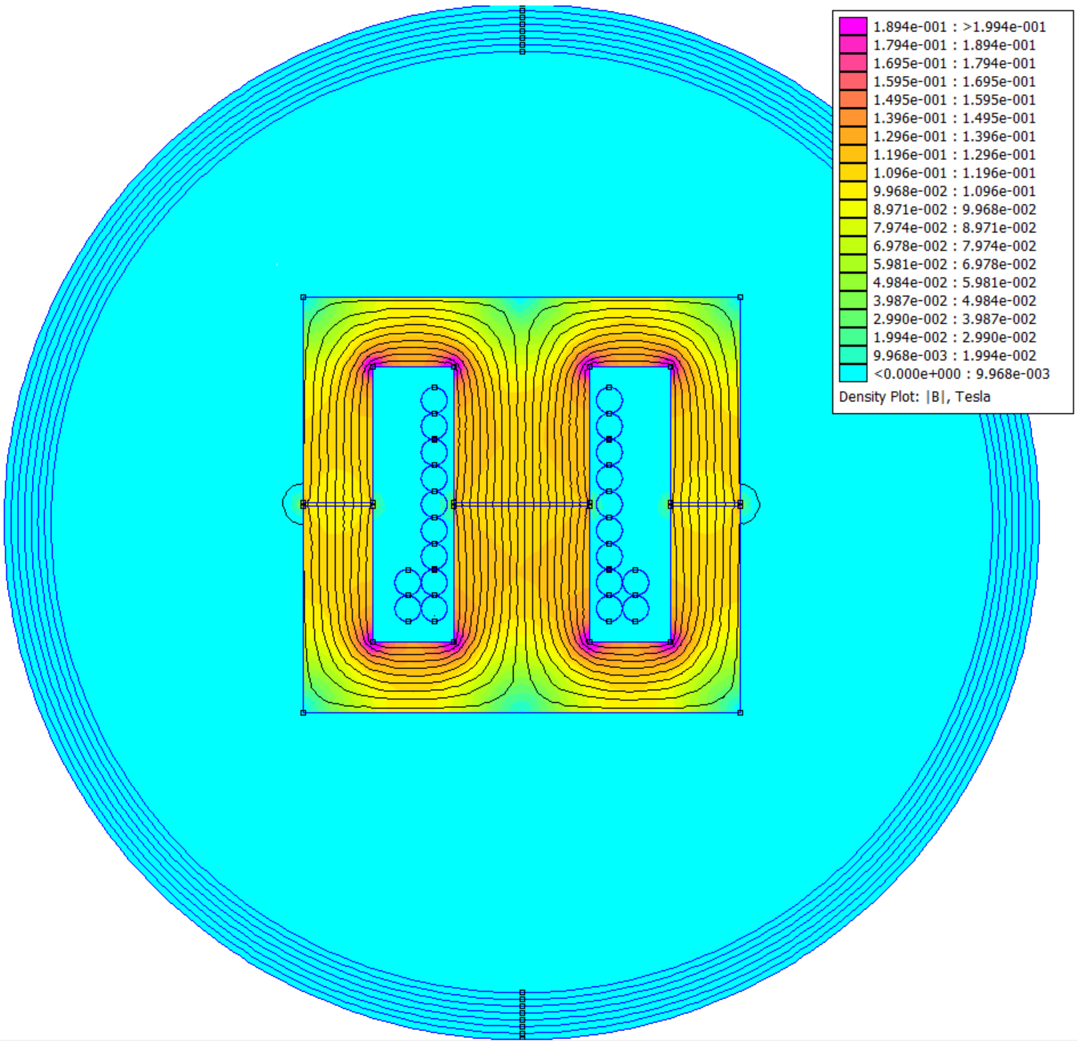
Once the installation finishes, a new message shows up and FEMM is ready to be used



7.7.2 Run simulation and get results

Once the model is generated, the user can run the simulation and get the desired results by pressing on "Launch FEMM".

The B field is automatically displayed upon completion:

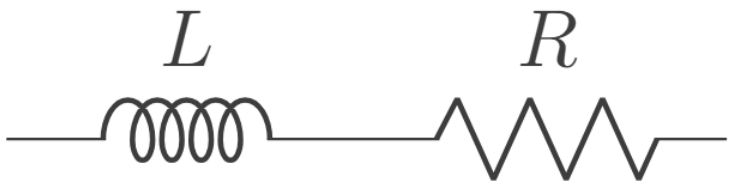


If the 'Generate reports' option was selected, some reports are automatically generated as well:

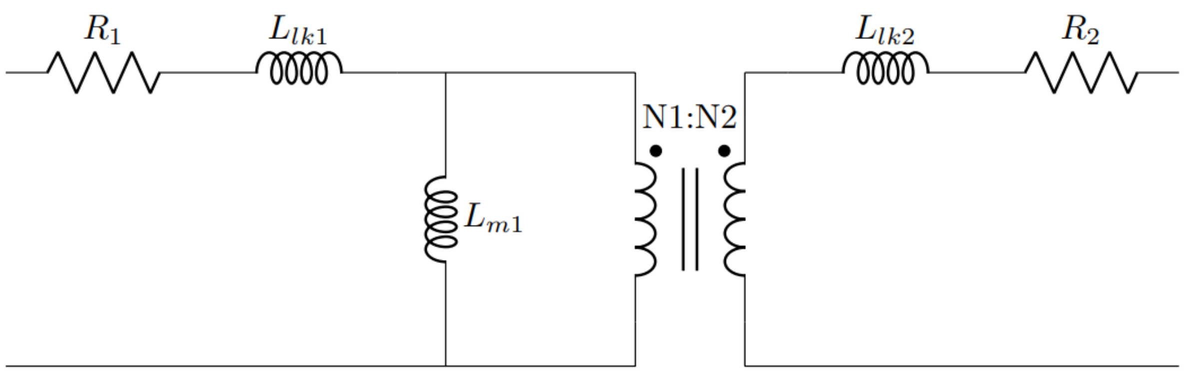
- For transformers:
  - Magnetizing inductance referred to primary (Lm1)
  - Primary leakage inductance (L_lk1)*
  - Primary resistance (R1)*
  - Secondary leakage inductance referred to secondary (L_lk2)*
  - Secondary resistance referred to secondary (R2)*
  - Primary self inductance (L11)
  - Secondary self inductance (L22)
  - Mutual inductance (L12)
  - Coupling coefficient
  - Effective turns ratio
- For inductors:
  - Inductance (L)

- Resistance (R)
- For both
  - Core loss
  - Winding loss

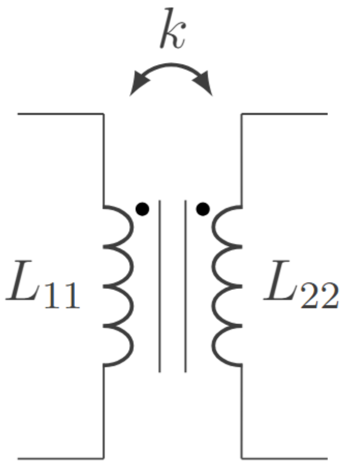
The equivalent circuit, constructed from the parameters given by the simulation of an inductor is show in the next figure (in the current version, core loss are not included in the inductor model):



The equivalent circuit, constructed from the parameters given by the simulation of a transformer is show in the next figure (in the current version core losses are not included in the transformer model):



A transformer can also be represented as two coupled inductors, defined by their inductance matrix:



$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} L_{11} & L_{12} \\ L_{21} & L_{22} \end{bmatrix} \frac{d}{dt} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix}$$

where the coupling coefficient "k" is defined as:

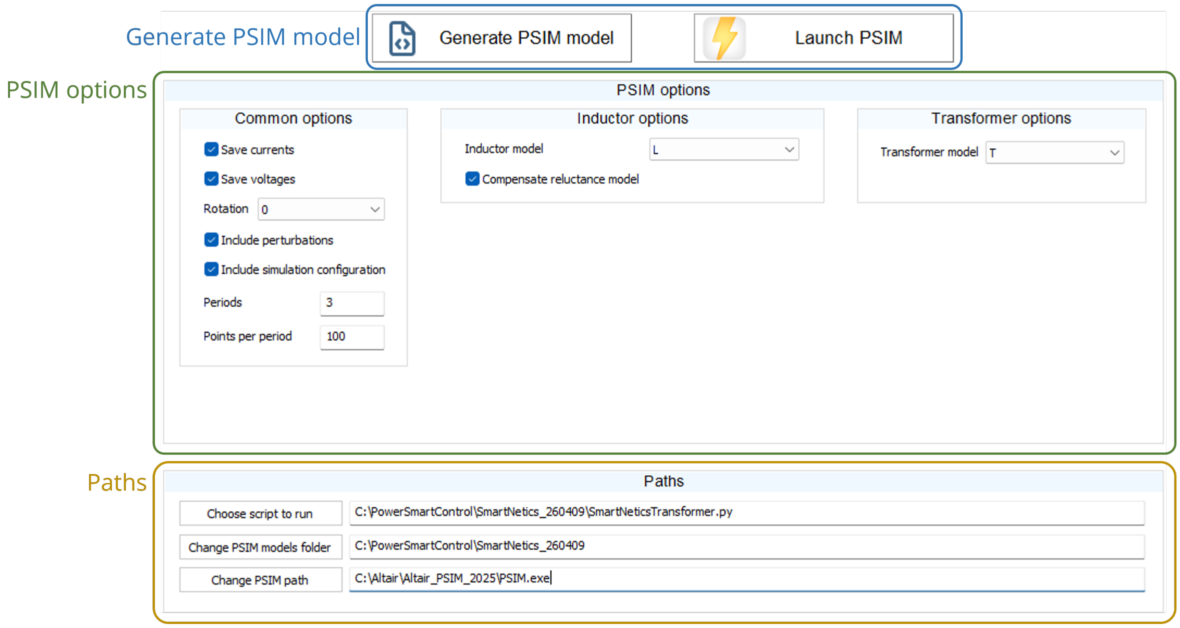
$$k = \frac{L_{12}}{\sqrt{L_{11} \cdot L_{22}}}$$

*In 2 simulations, on top of the simulated values, a correction is given to increase precision. That correction takes into account the difference between 2D and 3D simulations, allowing fast 2D simulation that provides a result matching the one of a 3D one.

7.8 PSIM

This tab allows the creation of the PSIM model of the selected device.
The model is generated as a python script, for which a PSIM installation is not needed. To build the model and run the simulation, an installation of Altair-PSIM is needed.

It consists of 3 panels:



- **Generate PSIM model**: generates the python script that is to be interpreted in PSIM. It can be opened in PSIM (in the machine that generated it or in any other machine with a PSIM installation).
- **Launch PSIM**: launches PSIM and automatically runs the generated script. The script to run can be manually set by the "Choose script to run" button below. PSIM has to be installed in the machine in the path selected below.

- **PSIM options**: select the options for the building of the model and the simulation configuration.:
  - **Common options**
    - **Save currents**: activate the option to automatically save the currents (current flag) in the components that allow doing so.
    - **Save voltages**: activate the option to automatically save the voltages (voltage flag) in the components that allow doing so.
    - **Rotation**: rotation of the device, in degrees.
    - **Include perturbations**: include a current source (and a voltage source at the secondary for transformers), connected to the component, that reproduce the waveforms used for the design.
    - **Include simulation configuration**: include a "Simulation Control" component in the schematic to configure the simulation. Checking this option enables 2 additional parameters:
      - **Periods**: number of periods of the waveform used as input to be simulated (adjusts the field 'Total Time' in 'Simulation Control').

- **Points per period**: number of points per period of the waveform to be simulated (adjusts the field 'Time Step' in 'Simulation Control').

- **Inductor options**
  - **Inductor model**: choose the PSIM model to be used to reproduce the inductor, a detailed description is given in the corresponding section. Available options are:
    - **L**: ideal model with a constant inductance, no losses or saturation effects.
    - **L_2**: model including the device equivalent resistance (taking winding and core losses into account).
    - **L_sat**: model not including resistance but taking into account the change in inductance with current.
      - *From PSIM documentation: Note that in certain situations, circuits that contain saturable inductors may fail to converge. In such a case, connecting a very small capacitor across the saturable inductor will help the convergence.
    - **L_mag_lin**: model built using the magnetic components found in PSIM, including winding (with equivalent resistance), gap (if any) and linear core (constant permeability and no loss).
  - **Compensate reluctance model**: Compensate the difference in gap needed to achieve a particular inductance depending on the model chosen for air reluctance in the design (PSIM always uses classical model). This option only applies to L_mag_lin components and is in its corresponding section.
- **Transformer options**
  - **Transformer model**: choose the PSIM model to be used to reproduce the transformer, a detailed description is available in the corresponding section. Available options are:
    - **T**: ideal model not including losses, inductances or saturation.
    - **T_1F**: model that includes inductance and resistance of each winding and magnetizing inductance.
    - **T_mag_lin**: model built using the magnetic components found in PSIM, including windings (with equivalent resistances) and linear core (constant permeability and no loss).

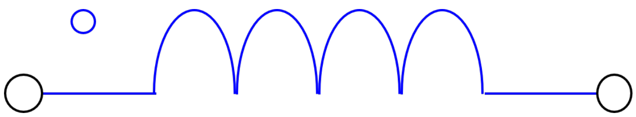
- **Paths**: configure the options regarding the paths.
  - **Choose script**: choose script to run if the "Launch PSIM" button is pressed.
  - **Change PSIM models folder**: choose the folder in which the models are going to be generated.
  - **Change PSIM path**: set the patch for the PSIM executable (PSIM.exe).

7.8.1 Inductors

The models currently available to reproduce an inductor are:

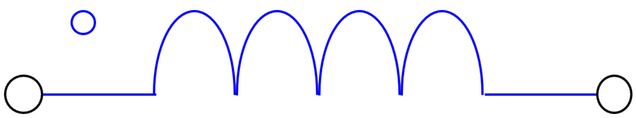
L

Ideal model with a constant inductance, no losses or saturation effects.



L_2

Model including the device equivalent resistance (taking winding and core losses into account).

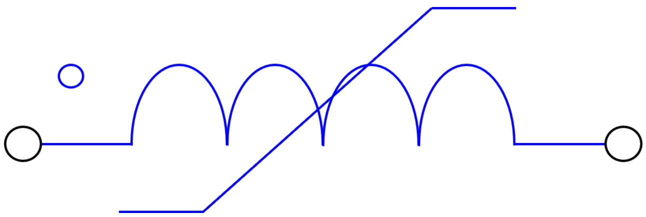


The equivalent resistance is calculated from the total loss (winding + core) estimated in SmartNetics and the RMS value of the inductor current:

$$R_{eq} = \frac{P_w + P_c}{I_{RMS}^2}$$

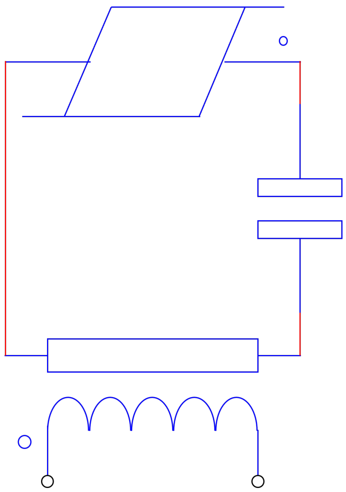
L_sat

Model not including resistance but taking into account the change in inductance with current.



L_mag_lin

Model built using the magnetic components available in PSIM, including winding (with equivalent resistance), gap (if any) and linear core (constant permeability and no loss).



The equivalent resistance of the winding is calculated from the winding loss estimated in SmartNetics and the RMS value of the inductor current:

$$R_{w,eq} = \frac{P_w}{I_{RMS}^2}$$

* If option "Compensate reluctance model" is not active, the gap length in the gap component is the same as the calculated one. If an air gap reluctance model different from the Classical one is selected, inductance may differ from calculations and experimental results for big gaps.

If it is active, the gap length is calculated so as to make inductance match with the one predicted by the air gap reluctance model selected for the design. Air gap length will only match the one displayed in SmartNetics if Classical model is selected for its calculation, since that is the one used in PSIM.

7.8.1.1 Compensate reluctance model

As described in the air reluctance section, if the fringing field around the gap is considered, an increase in the precision is achieved when calculating inductance for a gapped device

The classical expression, that does not consider this effect, may suffice for very small gaps, but gives a very high error when they increase. For this reason, more accurate models are present in SmartNetics.

When exporting to PSIM, though, only the classical expression is available:

$$\mathcal{R}_g = \frac{g}{\mu_0 \cdot w \cdot t}$$

To compensate for this phenomenon, an option is given to the user, in the check-box "Compensate reluctance model". When active, a new *equivalent gap* is calculated so that, even if the regular expression is used, the real inductance, considering the fringing flux, is achieved.

When exporting the circuit to PSIM, the same model used for the design, selected in the configuration dialog, is called and, from it, a new value for the gap, g', is calculated, so the inductance obtained in PSIM is:

$$\mathcal{R}_g = \frac{g'}{\mu_0 \cdot w \cdot t}$$

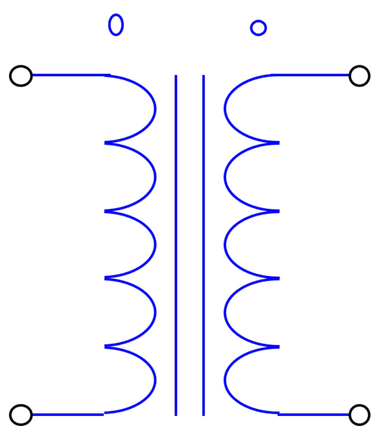
which will match the expected one.

7.8.2 Transformers

The models currently available to reproduce a transformer are:

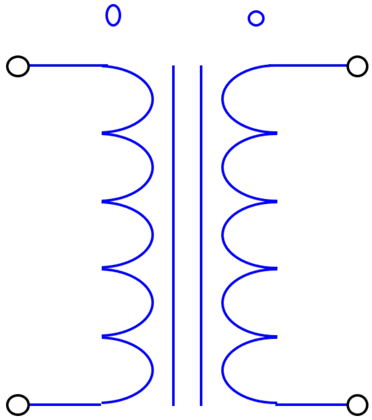
T

Ideal model not including losses, inductances or saturation.



T_1F

Model that includes inductance and resistance of each winding and magnetizing inductance.

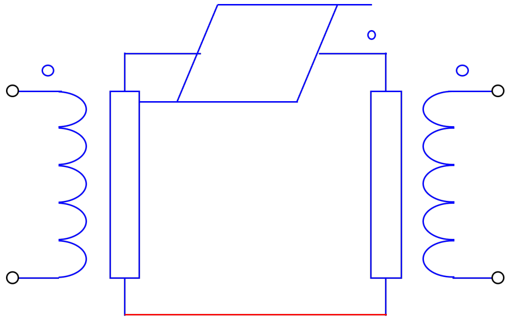


The equivalent resistance of the winding is calculated from the winding loss estimated in SmartNetics and the RMS value of the inductor current:

$$R_{w1,eq} = \frac{P_{w1}}{I^2_{1,RMS}} \qquad R_{w2,eq} = \frac{P_{w2}}{I^2_{2,RMS}}$$

T_mag_lin

Model built using the magnetic components found in PSIM, including windings (with equivalent resistances) and linear core (constant permeability and no loss)



The equivalent resistance of the winding is calculated from the winding loss estimated in SmartNetics and the RMS value of the inductor current:

$$R_{w1,eq} = \frac{P_{w1}}{I^2_{1,RMS}} \qquad R_{w2,eq} = \frac{P_{w2}}{I^2_{2,RMS}}$$

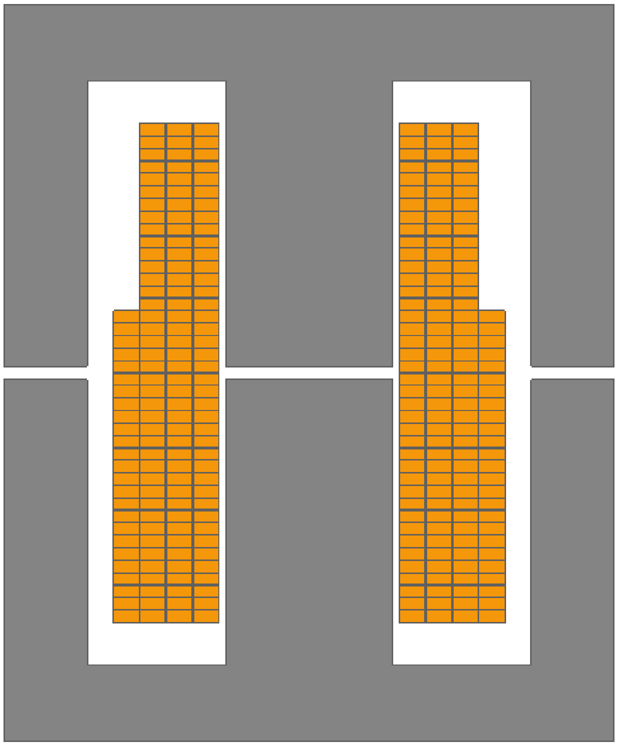
7.9 Common topics

There are some topics common to some of the different tabs:

- Improvement in 2D simulations

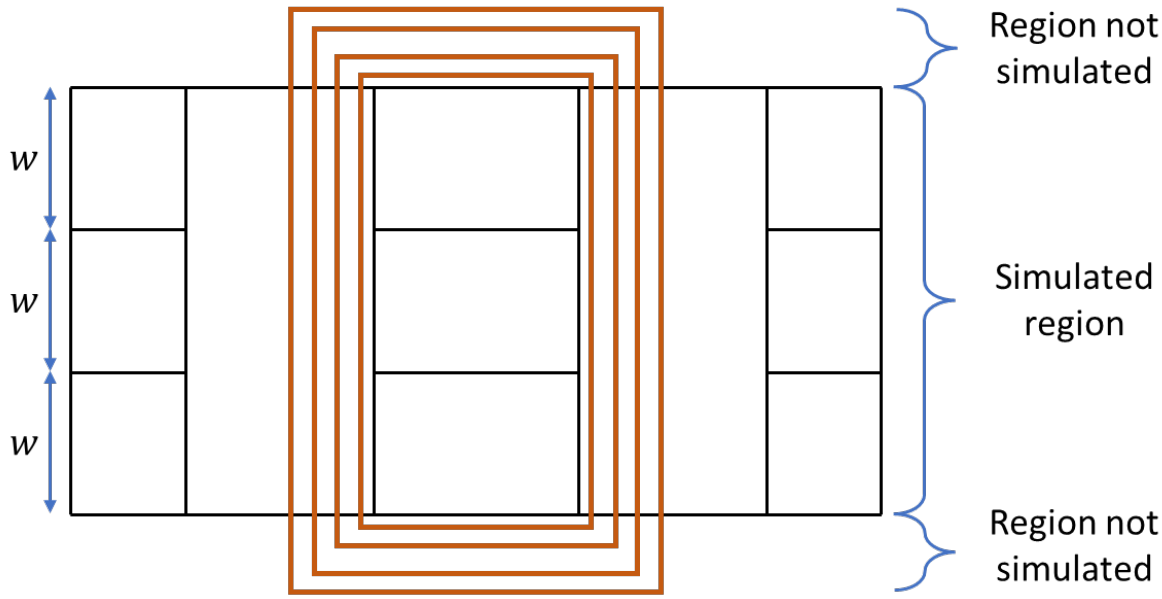
7.9.1 2D models

A 2D simulation can produce inductance values very close to the ones obtained in a 3D simulation, but spending only a fraction of time and resources.



Although the inductance due to the core presence is close to the one obtained in the 3D simulation, due to the fact that the core depth is taken into account in the calculations, the leakage and resistance values can be far from correct. This is due to the fact that the wire does not only extend to the length of the core, but it also needs to close at the front and back.

This means that, central legs that are not much longer than wider (which is the most common case), a very big part of the wire length is left out of the calculation. An example can be seen in the next figure, for a device with 3 stacked cores:



As can be seen, there is a portion of the device that is left out of the simulation, which means there is some length of conductor that is not taken into account.

To compensate that difference, the simulated resistances and leakage inductances are multiplied by the factor of the real length over the one used in simulation:

$$R_{compensated} = R_{simulated} \cdot \frac{length_{real}}{length_{simulated}}$$

$$L_{lk,compensated} = L_{lk,simulated} \cdot \frac{length_{real}}{length_{simulated}}$$

Thanks to this, a great increase in precision is achieved while maintaining very short simulation times by the 2D simplification.