Transformer cooling design and validation

Tutorial - May 2026

How to Contact:
info@powersmartcontrol.com
www.powersmartcontrol.com

Notice

Power Smart Control 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.

Disclaimer

Power Smart Control S.L. (PSC) makes no representation or warranty with respect to the adequacy or accuracy of this documentation or the software which it describes. In no event will PSC 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 and all other commercial damages or losses, or for any damages in excess of the list price for the license to the software and documentation.

Contents

Introduction

SmartNetics is a software for the design and analysis of magnetic devices: inductors and transformers. Our approach is not to offer a single one-size-fits-all solution, but to provide every possible design that matches a specification, along with an intuitive graphical interface. This combination allows the user to easily assess the impact of each value.

In previous tutorials (available at www.powersmartcontrol.com), we have shown how to design a dedicated magnetic component or how to analyze one that is already manufactured. In this case, we will combine both capabilities; we will **design a device for a given specification and cooling option, and then analyze how it would behave under a different ventilation configuration**.

This tutorial aims to illustrate how to tackle the design of a transformer under a given specification of voltage, current, and cooling, and then analyze how it is going to behave under a different one. On top of the analytical calculations performed inside SmartNetics, this tutorial illustrates how to export the design to third-party tools for validation.

The transformer we are going to design needs to operate under the following restrictions:

Voltage	Current	Frequency	Turns ratio	Geometry	Material
350 V (peak)	30 A (peak)	50 kHz	1:1	E55/28/21	3C94

Table 1: Main design parameters

First, we will design the transformer to operate under natural convection, and then we are going to check how we can improve it by including a fan.

Transformer design

1 Input data

As stated in the previous section, the restrictions for the design of the transformer, imposed by the rest of the circuit, are:

- Device: Transformer.
- Turns ratio: 1:1.
- Operation frequency: 50 kHz.
- Current: sinusoidal 30 A (Amplitude).
- Voltage: sinusoidal 350 V (Amplitude).
- Core geometry: E55/28/21.
- Core material: 3C94.
- Cooling: natural convection.

The first parameters, "Turns ratio" and "Operation frequency" can be set on the top bar of the first dialog ("Input data"), once "Transformer" has been selected in the top switch, as shown in Figure 1.

Figure 1: Inductance and frequency

For a transformer, the primary side current and voltage waveforms are needed. They can be described right below the “Turns ratio” and “Frequency” parameters, as shown in Figure 2.

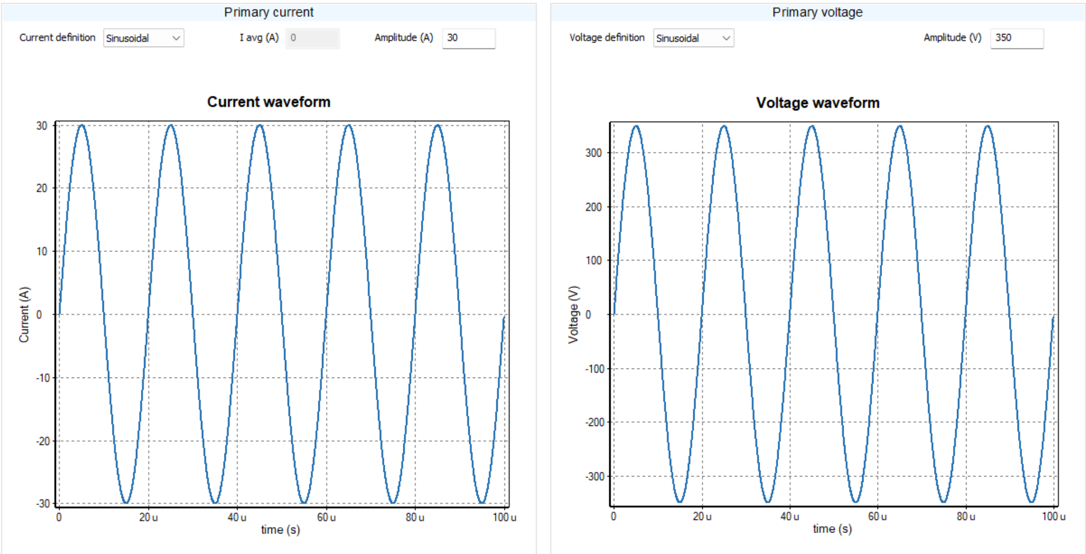
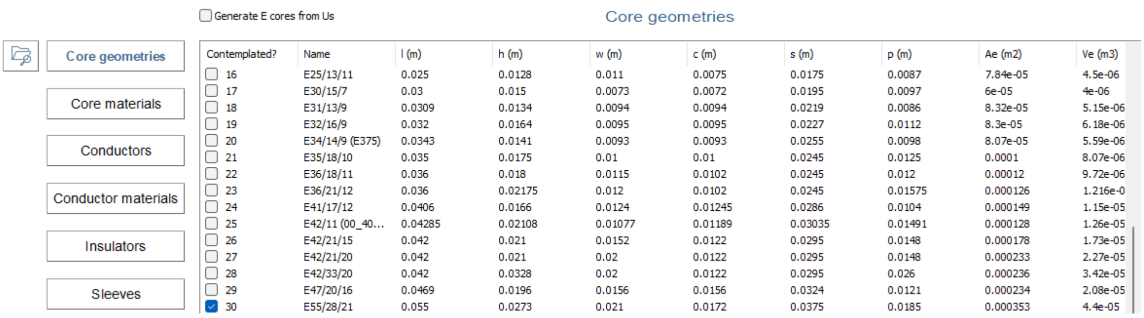


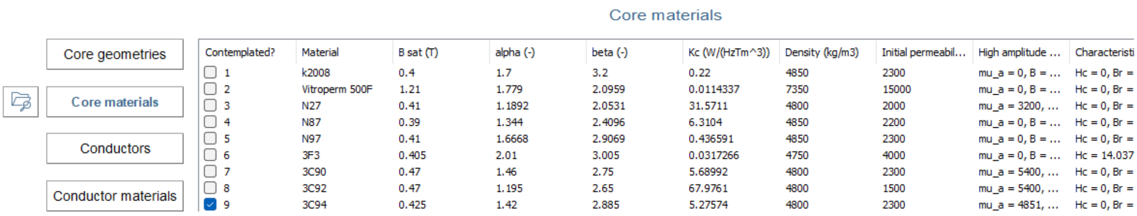
Figure 2: Primary current and voltage

For this transformer, we are restricted to a given core material and geometry: cores E55/28/21 of 3C94 by Ferroxcube. We can apply that restriction by activating only those particular entries in the database. This is done in the second dialog (“Configuration”) in the “Databases” tab, as shown in the next figures:



Contemplated?	Name	l (m)	h (m)	w (m)	c (m)	s (m)	p (m)	Ae (m2)	Ve (m3)
☐ 16	E25/13/11	0.025	0.0128	0.011	0.0075	0.0175	0.0087	7.84e-05	4.5e-06
☐ 17	E30/15/7	0.03	0.015	0.0073	0.0072	0.0195	0.0097	6e-05	4e-06
☐ 18	E31/13/9	0.0309	0.0134	0.0094	0.0094	0.0219	0.0086	8.32e-05	5.15e-06
☐ 19	E32/16/9	0.032	0.0164	0.0095	0.0095	0.0227	0.0112	8.3e-05	6.18e-06
☐ 20	E34/14/9 (E375)	0.0343	0.0141	0.0093	0.0093	0.0255	0.0098	8.07e-05	5.59e-06
☐ 21	E35/18/10	0.035	0.0175	0.01	0.01	0.0245	0.0125	0.0001	8.07e-06
☐ 22	E36/18/11	0.036	0.018	0.0115	0.0102	0.0245	0.012	0.00012	9.72e-06
☐ 23	E36/21/12	0.036	0.02175	0.012	0.0102	0.0245	0.01575	0.000126	1.216e-0
☐ 24	E41/17/12	0.0406	0.0166	0.0124	0.01245	0.0286	0.0104	0.000149	1.15e-05
☐ 25	E42/11 (00_40…	0.04285	0.02108	0.01077	0.01189	0.03035	0.01491	0.000128	1.26e-05
☐ 26	E42/21/15	0.042	0.021	0.0152	0.0122	0.0295	0.0148	0.000178	1.73e-05
☐ 27	E42/21/20	0.042	0.021	0.02	0.0122	0.0295	0.0148	0.000233	2.27e-05
☐ 28	E42/33/20	0.042	0.0328	0.02	0.0122	0.0295	0.026	0.000236	3.42e-05
☐ 29	E47/20/16	0.0469	0.0196	0.0156	0.0156	0.0324	0.0121	0.000234	2.08e-05
☑ 30	E55/28/21	0.055	0.0273	0.021	0.0172	0.0375	0.0185	0.000353	4.4e-05

Figure 3: Core geometries database



Contemplated?	Material	B sat (T)	alpha (-)	beta (-)	Kc (W/(HzTm^3))	Density (kg/m3)	Initial permeabil…	High amplitude …	Characteristi
☐ 1	k2008	0.4	1.7	3.2	0.22	4850	2300	mu_a = 0, B = …	Hc = 0, Br =
☐ 2	Vitroperm 500F	1.21	1.779	2.0959	0.0114337	7350	15000	mu_a = 0, B = …	Hc = 0, Br =
☐ 3	N27	0.41	1.1892	2.0531	31.5711	4800	2000	mu_a = 3200, …	Hc = 0, Br =
☐ 4	N87	0.39	1.344	2.4096	6.3104	4850	2200	mu_a = 0, B = …	Hc = 0, Br =
☐ 5	N97	0.41	1.6668	2.9069	0.436591	4850	2300	mu_a = 0, B = …	Hc = 0, Br =
☐ 6	3F3	0.405	2.01	3.005	0.0317266	4750	4000	mu_a = 0, B = …	Hc = 14.037
☐ 7	3C90	0.47	1.46	2.75	5.68992	4800	2300	mu_a = 5400, …	Hc = 0, Br =
☐ 8	3C92	0.47	1.195	2.65	67.9761	4800	1500	mu_a = 5400, …	Hc = 0, Br =
☑ 9	3C94	0.425	1.42	2.885	5.27574	4800	2300	mu_a = 4851, …	Hc = 0, Br =

Figure 4: Core materials database

For the wire, we do not have such restrictions, so we will consider two options: a Rigid wire and a Litz one. With the aim to ease manufacturing, we will activate the "Force same conductor" option, so primary and secondary use the same wire. The configuration is shown in the Figure 5.

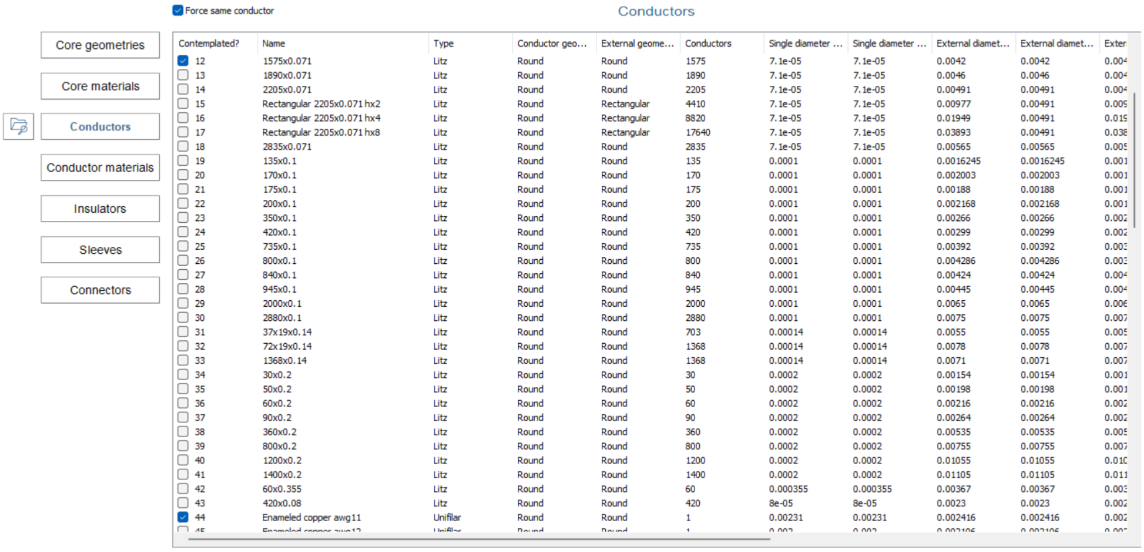


Figure 5: Wire geometries database

Even though we are restricted to a given core geometry, there is still freedom regarding the use of several stacked cores. We will allow that possibility, but restrict it to 4 stacked cores as a maximum. That option can be selected in the "Device parts" tab, as shown in Figure 6.

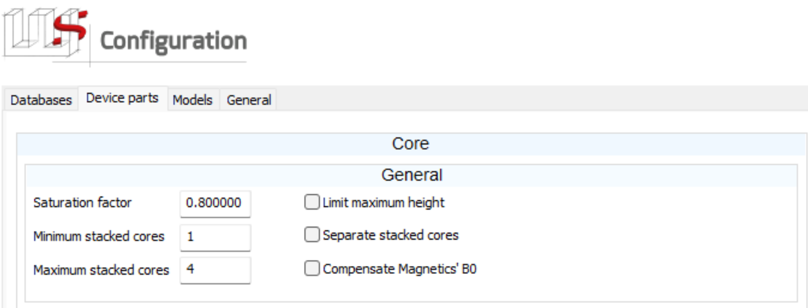


Figure 6: Maximum stacked cores

Last, we can configure the cooling strategy for the device. For the first design iteration, we will impose natural convection by ensuring the "Fan cooling" checkbox is not active, and we will allow a maximum temperature of 250 °C. Those options are configured in the "Models" tab, which will be as shown in Figure 7.

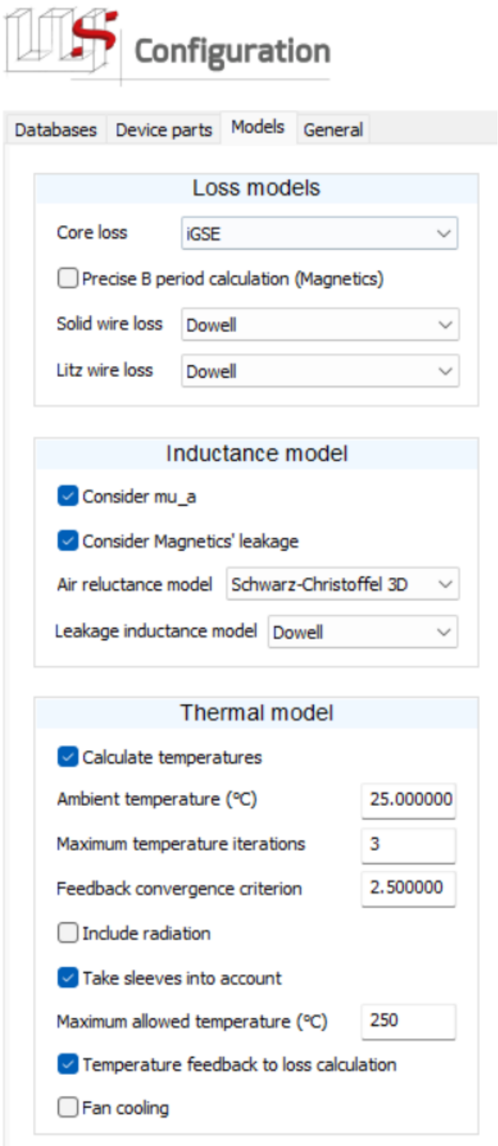


Figure 7: Models configuration

We will leave everything else as configured by default, since it can provide good results for most specifications.

Now, instead of getting a single black-box design, we will calculate every possible combination of the items in the databases that provides a valid solution (while complying with the imposed restrictions). To do so, we need to go to the next dialog: "Design".

2 Design

By default, the 4 axes will be empty, as shown in the next figure:

Figure 8: Empty design space

Once "Begin magnetic design" is pressed, the procedure begins and, once it finishes, the results are displayed in the 4 axes. The variables to be used can be chosen at the right side, and every numeric variable that has any impact on the design can be selected.

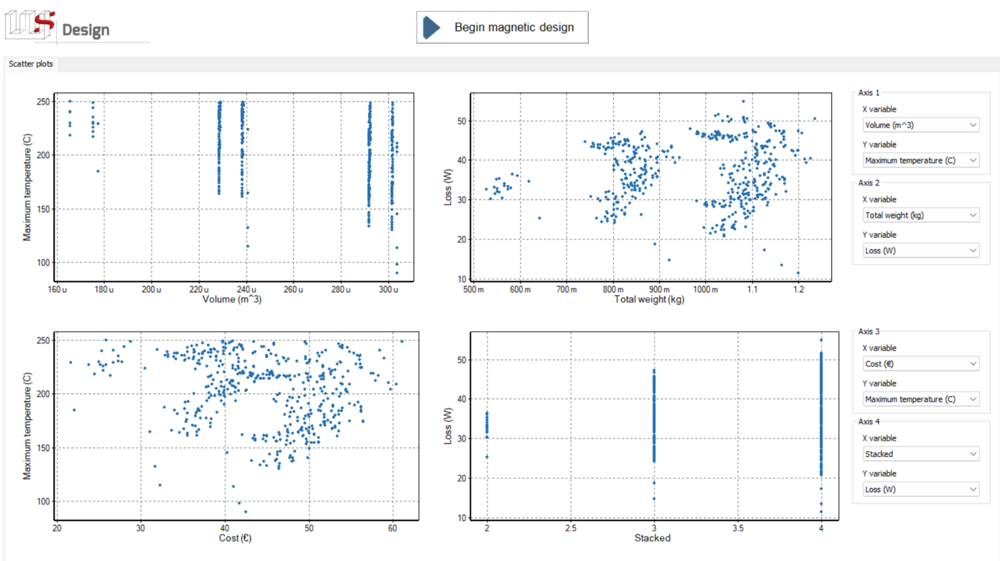


Figure 9: Available designs

From every available design (shown as every blue dot in the axes), the user can select a subset of those interesting for their needs. That can be done by clicking and dragging in any of the 4 graphs.

As can be seen in Figure 10, the selected devices are highlighted in orange in every axis. Thanks to this, the impact of a variable on the remaining seven can be easily analyzed.

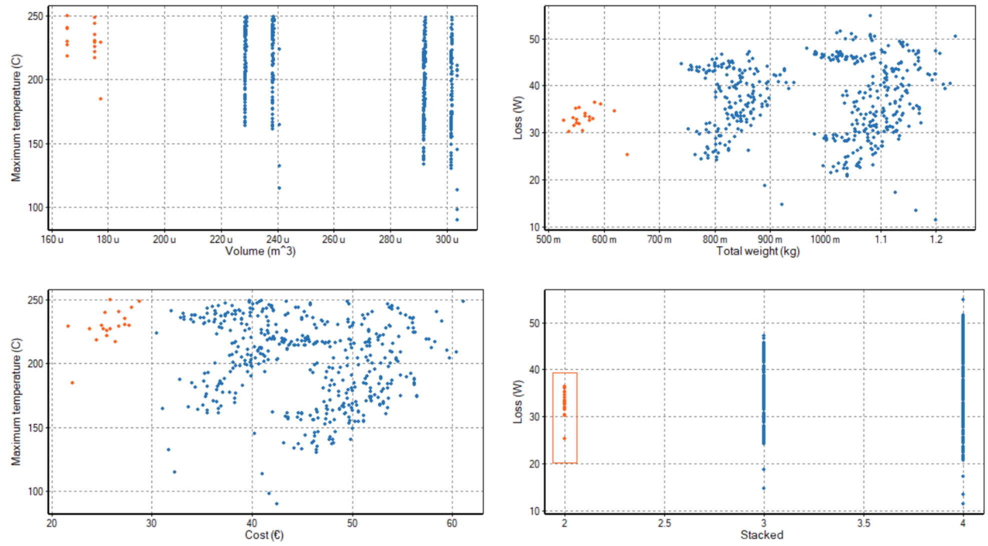


Figure 10: Available designs. Selecting single-core ones

For this design, we are going to select only the devices with a temperature lower than 100 °C, as shown in the next figure:

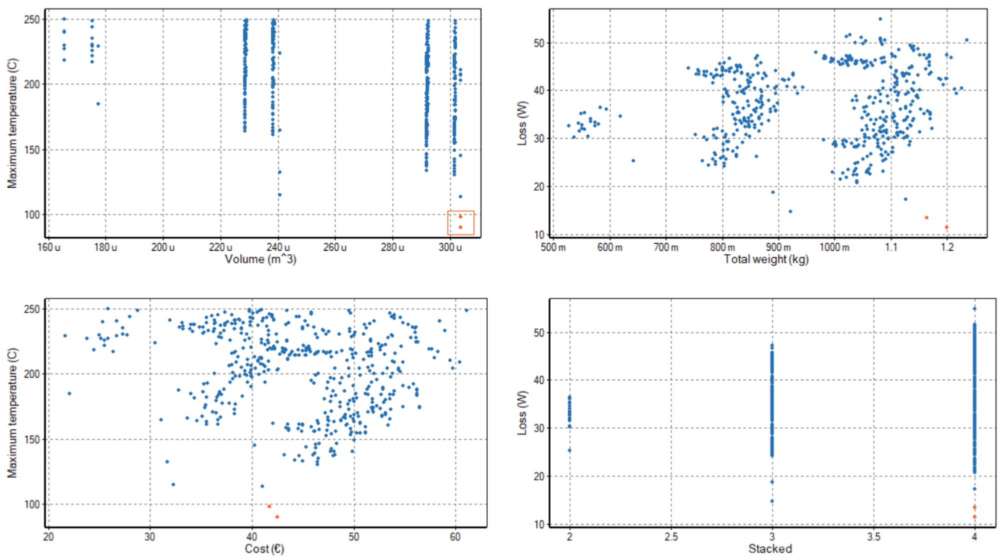


Figure 11: Available designs. Devices under 100°C

In the dialog shown in the previous figures, only 8 variables can be displayed at the same time. For a deeper analysis, we can access the next dialog, “Selection”, to have access to every detail of the selected devices, including inductances, losses (divided into core and winding), maximum temperature, maximum magnetic field, etc. Values that are not numeric (like core material) are also shown in the new dialog.

In this dialog, only the devices selected in the previous step are shown:

Selection

	Cores	Material	Stacked	N_1	N_2	Window usage	Primary window...	Primary wire	Secondary wire	Primary vertical...	Primary horizon...	Secondai
1	E55/28/21	3C94	4	7	7	0.649767	0.417973	1575x0.071	1575x0.071	1	1	1
2	E55/28/21	3C94	4	8	8	0.742591	0.417973	1575x0.071	1575x0.071	1	1	1

Figure 12: Selection dialog

With the scroll bar, the user can access the different parameters. In this case, we can see that the main difference between the two selected devices is their number of turns. That difference results in different wire losses and, above all, in different B field and core losses, as can be seen in Figure 13.

R_{DC;1}	R_{DC;2}	P_{w;1} (W)	P_{w;2} (W)	P_{c} (W)	Delta Bpp	l_w_1 (m)	l_w_2 (m)	Total weight (kg)	Volume (m^3)	Area (m^2)	Loss (W)	L_{mu} (H)	Maximum temperature (C)
0.00551941	0.00637449	2.51139	2.90045	8.04102	0.225432	1.5372	1.7752	1.16371	0.000303603	0.0055605	13.4529	0.00161267	98.2424
0.00617197	0.00712901	2.83249	3.27167	5.47022	0.197253	1.7568	2.0288	1.20011	0.000303603	0.0055605	11.5744	0.00210635	90.3635

Figure 13: Selection dialog. Losses and temperature details

Since the one with the lowest losses and temperature is the second one (with 8 turns), we will select that device. To do so, click on any cell of said design and then click on "Select design" at the bottom. The selected device will be reproduced at the bottom, as shown in Figure 14.

Select design

	Cores	Material	Stacked	Stacked cores d...	Insulator	Wiring	Inner winding	N_1	N_2	Window usage	Primary window...	Primary wire	Secondary wire	Primary vertical...	Primary
1	E55/28/21	3C94	4	0	NOMEX	Central	Primary	8	8	0.755851	0.442455	1575x0.071	1575x0.071	1	1

Figure 14: Single selected device

Now that a single device is selected, the user can access the last dialog, "Device", for a graphical view of the most important parameters, like geometry or temperature.

A 3D representation of the device and its parts can be seen in the "Geometry" tab:

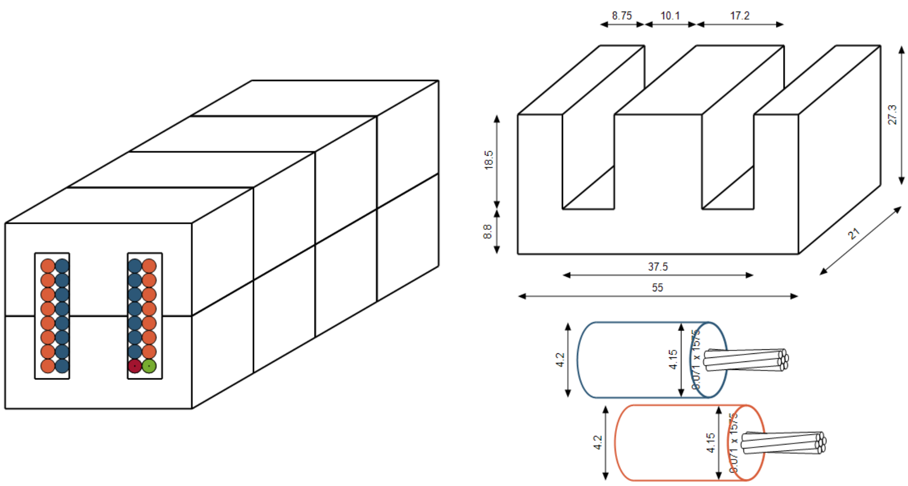


Figure 15: Original design geometry

The expected temperature, loss, weight, and cost distributions are presented in the "Performance" tab, as shown in Figure 16.

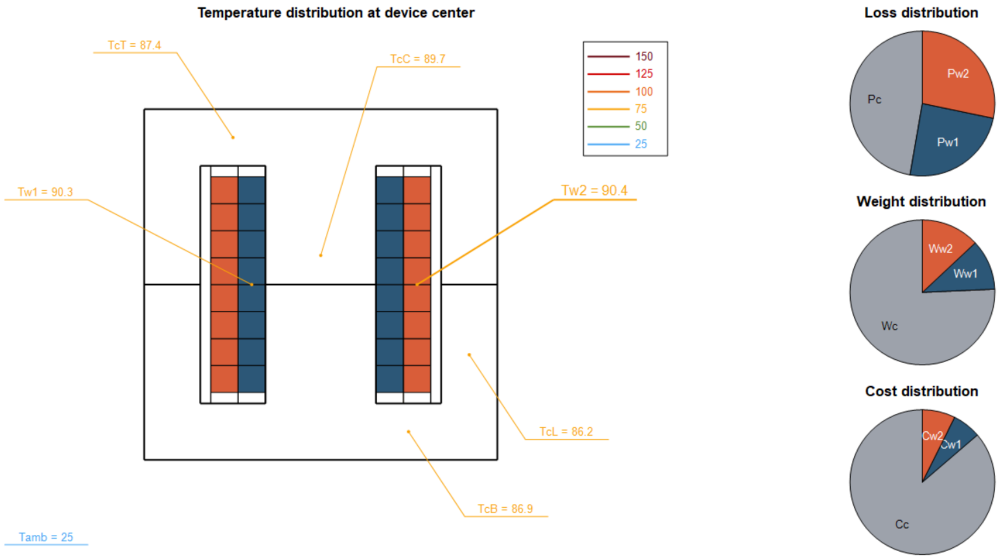


Figure 16: Original design temperature distribution

Up to this point, we have showcased one of the uses of SmartNetics: a design tool for a magnetic component. As shown, this is done by designing every possible combination of parameters that provide a valid solution, and letting the user select the one that is best for their project.

In the next sections of this tutorial, we will show the other use of SmartNetics: a tool for the analysis of a given magnetic component. Once the user has selected a device, they can set every design parameter to a fixed value in the configuration options. This way, a single solution is generated in the design procedure, and the tool can be used not to design, but to analyze said solution. The next chapter provides an example on how to carry out such analysis.

Transformer analysis and optimization

In SmartNetics, every combination of parameters that provides a valid result is designed. If every design parameter is set to a given value, there is a single device that can result from the design process, and the tool can be used to analyze that design.

1 Analysis

The values can be set in the second dialog, "Configuration", on the "General" tab, as shown in the next figure:

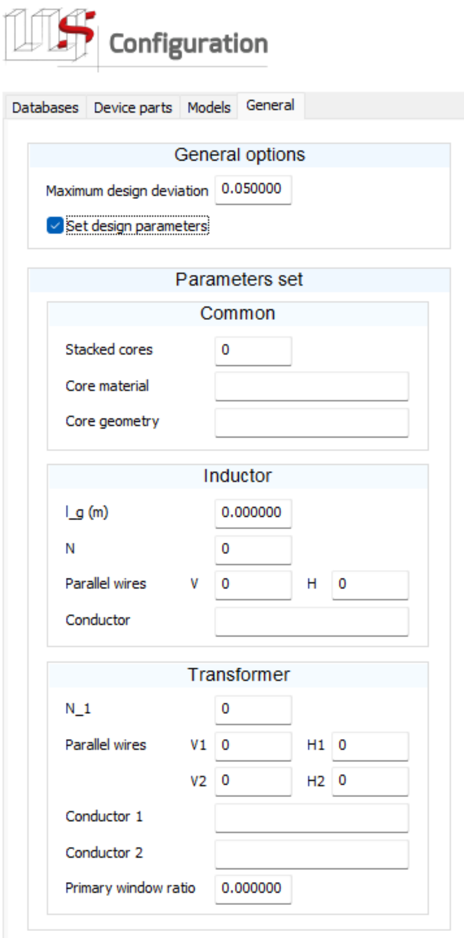


Figure 17: Configuration. Empty design parameters

By default, nothing is set, so every box is empty or set to 0. That means the tool will consider the value unknown and will look for any solution that fits the requirements.

Not every parameter has to be provided, the user can fix some values and let the tool calculate the unknowns.

In this case, we will only set some of them to the ones of the previously designed transformer. The configuration will look like this:

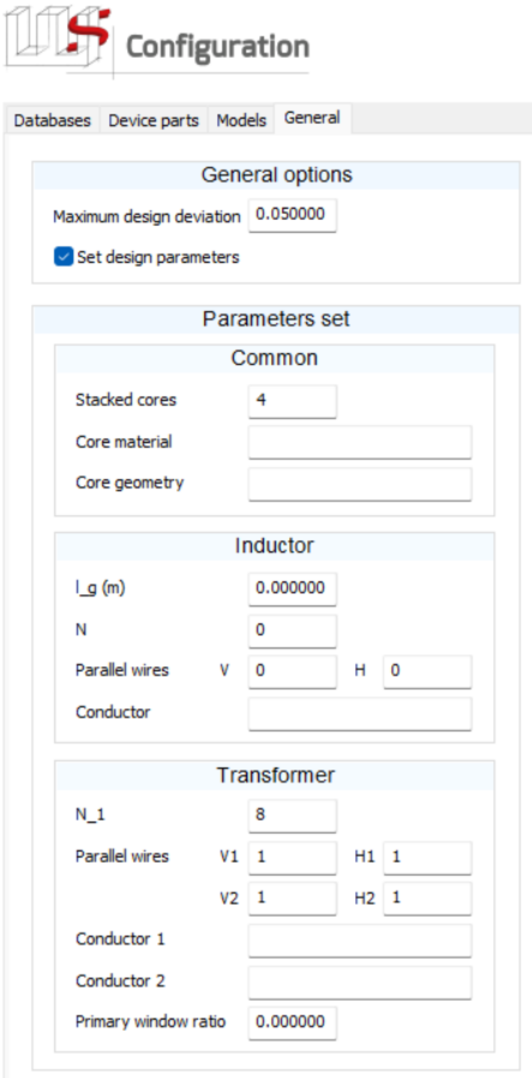


Figure 18: Configuration. Set design parameters

Now, if we run the design again, a single solution is given, corresponding to the device we selected before, as shown in Figure 19.

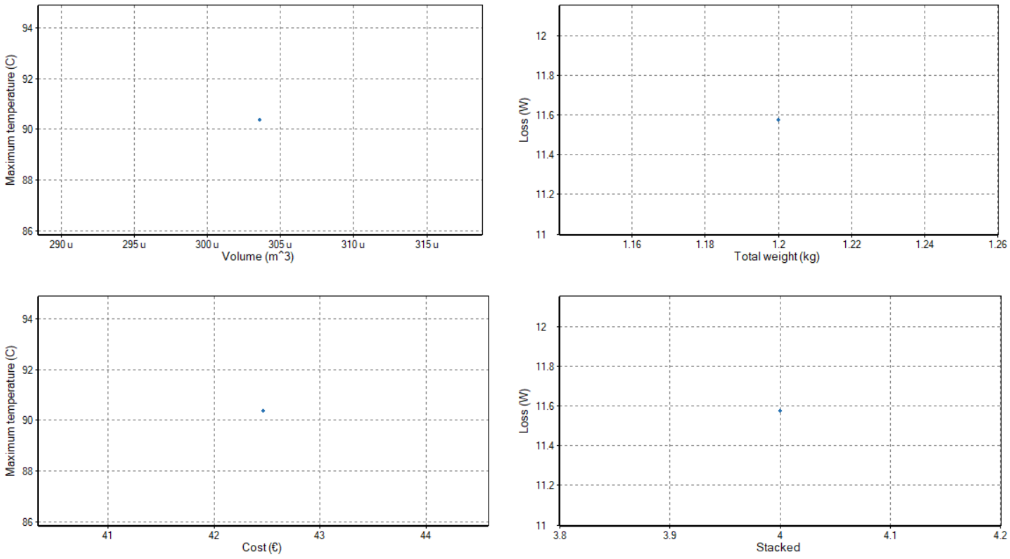


Figure 19: Single available device

Once a single device is possible, the user can analyze it to check how it is going to behave under different conditions, for example:

- Different voltage.
- Different current.
- Different cooling configuration.

In this particular example, voltage and current are fixed requirements, so we will leave them as they are, but we will modify the cooling options to assess their impact and to improve the behavior of our converter.

2 New cooling option

The cooling is configured in the “Configuration” dialog, on the “Models” tab. To improve the cooling, we will add a fan by activating the corresponding checkbox at the bottom of the screen. The full new configuration is shown in Figure 20.

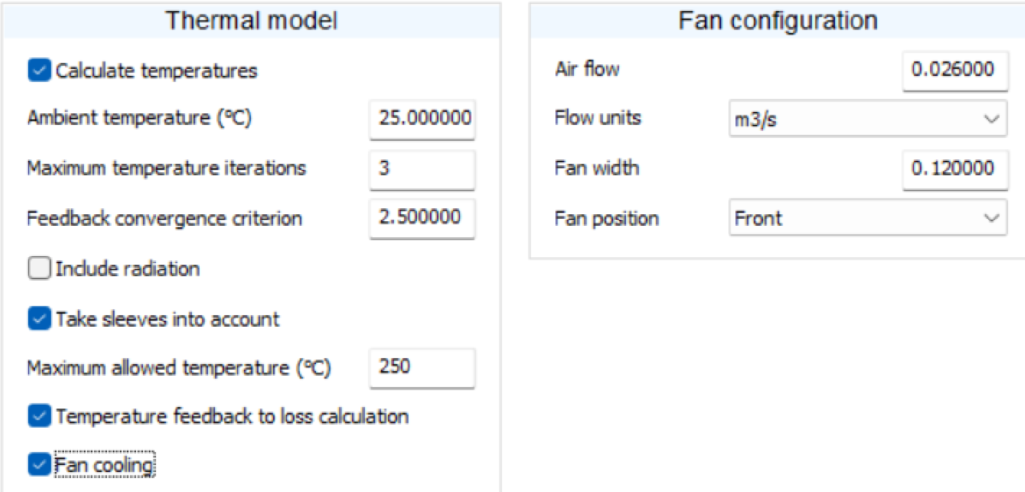


Figure 20: Fan configuration

Now, if we run the design procedure again, we will get the same device (same core, same materials, same turns, etc.), because everything is set to a given value. However, since the cooling option has changed, the temperature (and everything that depends on it) is going to change.

We can check it by running the design procedure again. As can be seen in Figure 21, there is only one device again, but the temperature is much lower (and losses have decreased as well, since they depend on temperature).

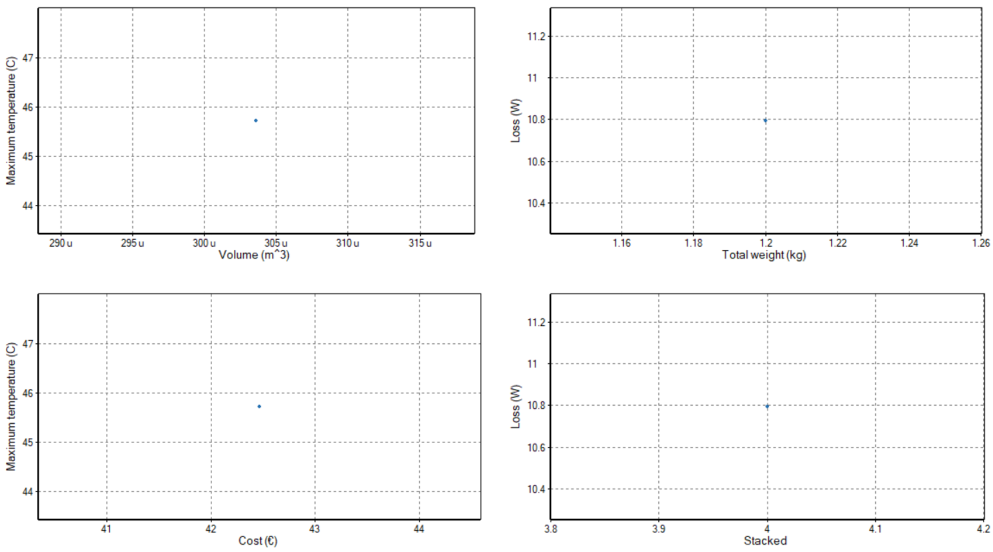


Figure 21: Single result for new cooling

In the “Device” dialog, in the “Geometry” tab, we can check that the device is still the same:

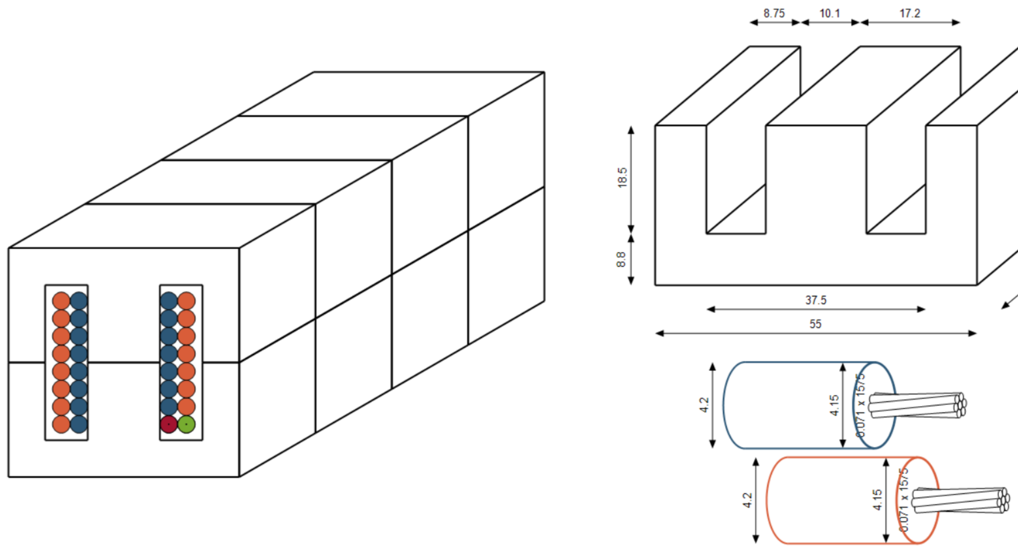


Figure 22: Original design geometry

And, in the “Performance” tab, we can check the new temperature distribution, reduced thanks to the use of a fan.

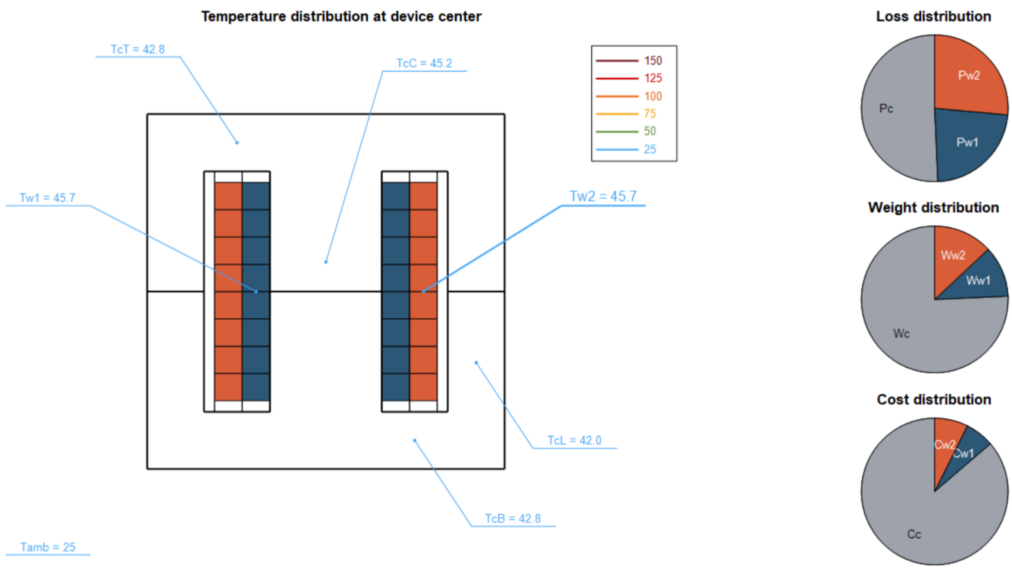


Figure 23: Original design temperature distribution

Up to this point, every result displayed has been obtained by means of analytical equations, which allows the very fast calculations of any number of possible combinations. Once the component has been redesigned and a single device has been selected, we can double-check its main parameters by exporting it to third-party tools. In this regard, SmartNetics provides three exporting options to Finite Element Analysis software: Ansys (Maxwell for electromagnetic and Icepak for thermal), Altair-Flux (only for electromagnetic), and FEMM (for 2D electromagnetic). On top of that, you can also export your device to PSIM to include it in your circuit simulation.

Validation

For this tutorial, we will perform three exports to third-party tools:

- FEMM for inductance and losses validation.
- Ansys-Icepak for thermal validation
- PSIM for circuit simulation

Each export has its own tab in the last dialog.

1 FEMM

The configuration used for the FEMM simulation is shown in Figure 24

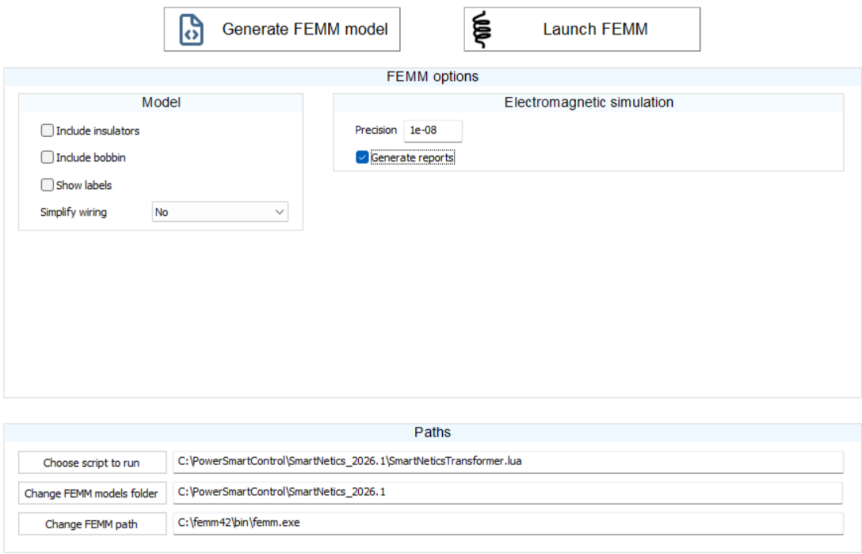


Figure 24: FEMM export configuration

The user can replicate the design selected in SmartNetics by clicking on the "Generate FEMM model" button. Once generated, it can be simulated on the same computer (if FEMM is installed) or on any other. If installed, the user can click on "Launch FEMM" to automatically launch the software, run the simulations, and display the results, as shown in Figure 25.

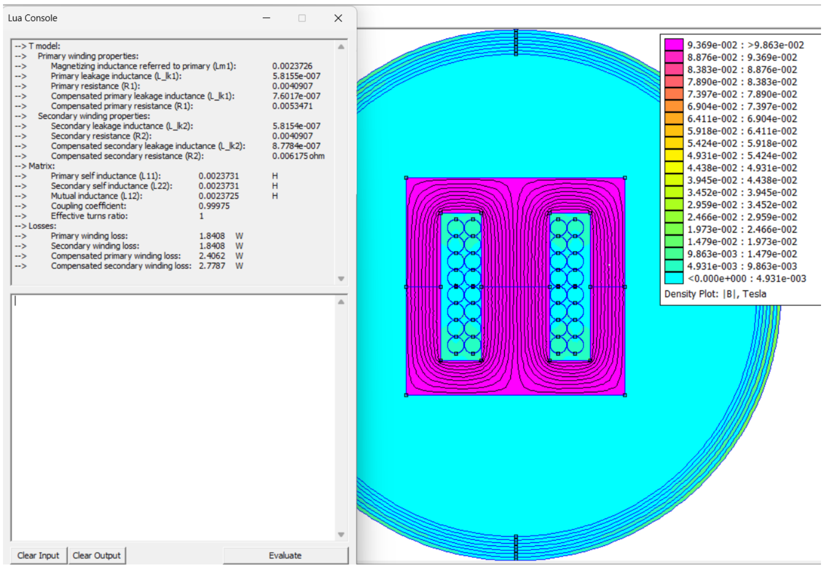
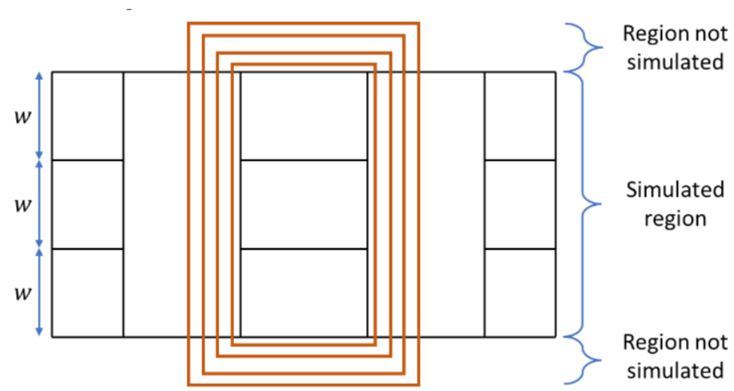


Figure 25: FEMM results

The use of a 2D simulator allows a very fast calculation, but also has some drawbacks. One of the drawbacks is the fact that the portion of the wire that is outside the core is not considered in the simulation. This affects both wire resistance and leakage inductance. SmartNetics automatically compensates for that, so the user can get results close to a 3D simulation with the speed of a 2D one. The compensation is explained in the SmartNetics help files and reproduced here:



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:

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

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

Figure 26: Wire length compensation

Once losses have been validated, we can proceed to export the model to Ansys-Icepak for a temperature validation.

2 Ansys Icepak

Using Ansys-Icepak, we can check the temperature distribution for the full volume of the device. To provide a good depiction of the real problem, we will include every component in the simulation. Apart from the core and winding, we will use the check-boxes to include insulators, wire sleeves, and bobbin, as shown in the next figure:

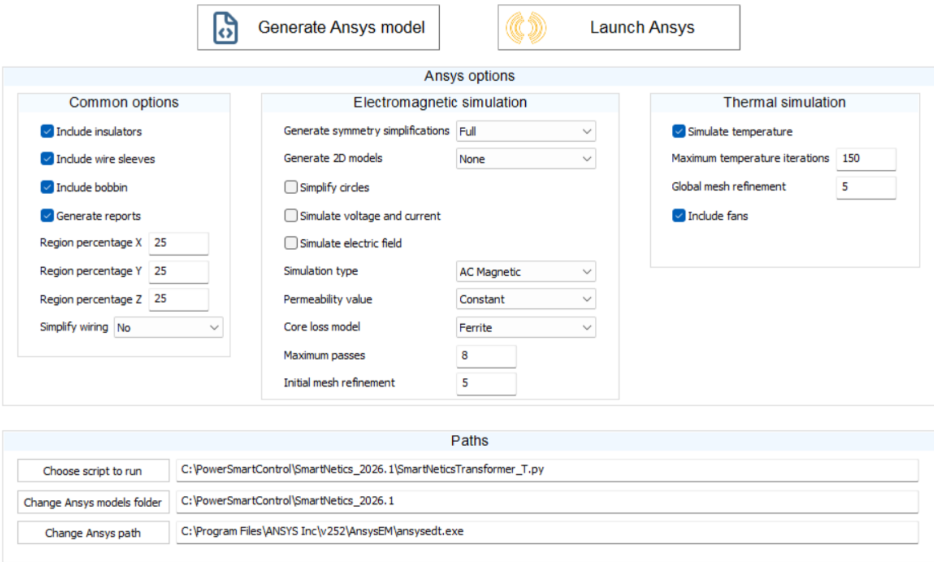


Figure 27: Ansys export configuration

As in FEMM, the user can generate the Ansys model by clicking on the "Generate Ansys model" button. Once generated, it can be simulated on the same computer (if Ansys is installed) or on any other. If installed, the user can click on "Launch Ansys" to automatically launch the software and build the model:

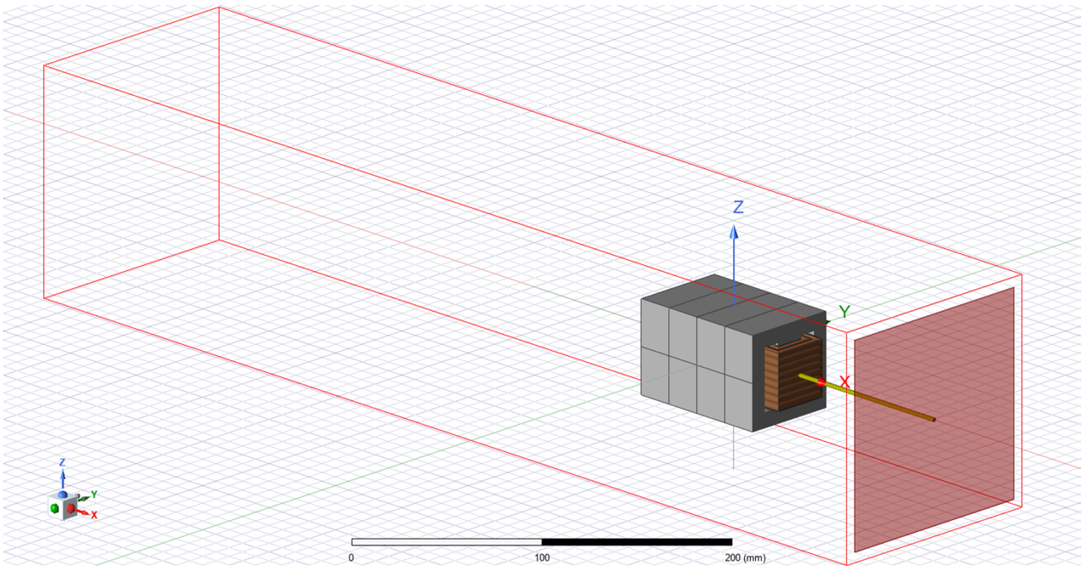


Figure 28: Icepak temperature results

Once in Ansys-Icepak, the user can click on "Analyze All" to run the simulation:

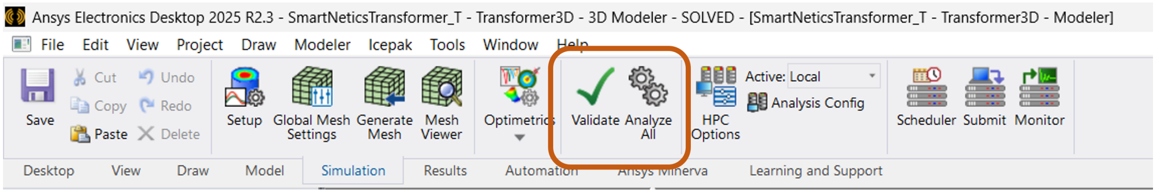


Figure 29: Ansys run button

After the simulation finishes, we can check the temperature distribution:

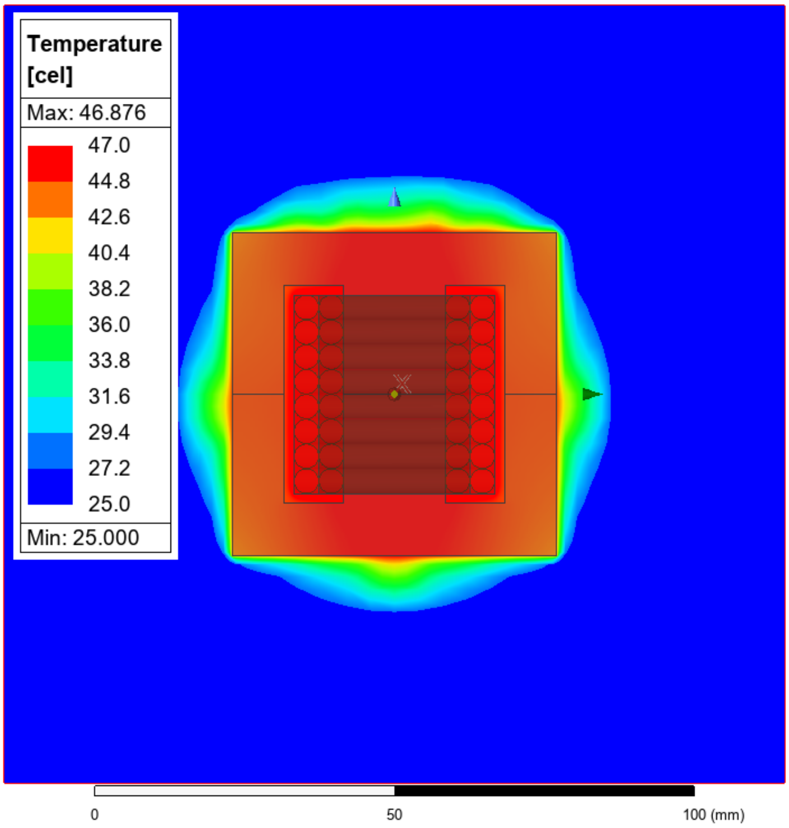


Figure 30: Ansys-Icepak temperature results

3 PSIM

Once we know the losses and temperature calculations are correct, we can export our design to PSIM to study its behavior or to include it in the simulation of the whole power converter.

As in the previous tools, the user can generate the model by clicking on the "Generate PSIM model" button. Once generated, it can be simulated in the same computer (if PSIM is installed) or in any other. If installed, the user can click on "Launch PSIM" to automatically launch the software and build the model.

The user can select how to export their inductor or transformer from the options present in PSIM (and described in the SmartNetics help file, accessed by pressing F1). For example, to export a transformer model that includes inductances and wire resistances, the model "t_1F is selected".

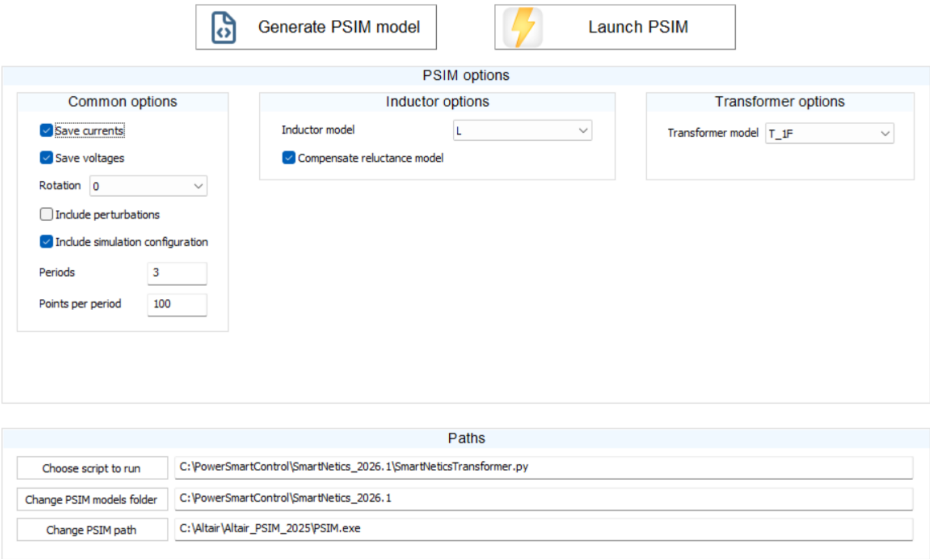


Figure 31: PSIM export configuration

Once in PSIM, the device parameters can be checked, as shown in the next figures.

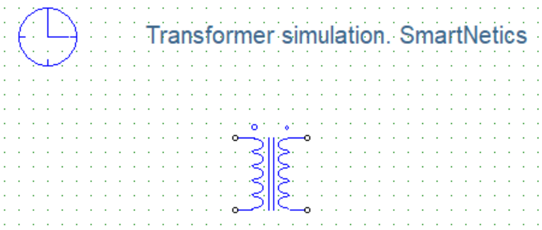


Figure 32: PSIM transformer device

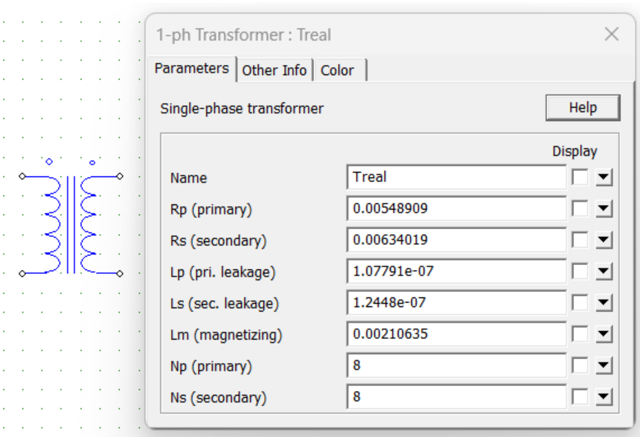


Figure 33: PSIM transformer values

Since all the information is included in the component itself, it can easily be used in any circuit.

Conclusions

In this tutorial, we have shown how to design a transformer for a given specification and how to analyze its behavior under a different one.

Thanks to SmartNetics capabilities, we have been able to design a device able to operate without any cooling and then to assess how its behavior is going to change if a fan is used.

Last but not least, the user can automatically export the full model to third-party tools for its simulation, including Ansys (Maxwell or Icepak), Altair-Flux, FEMM, and PSIM.

This tutorial is intended as an example, so the user is encouraged to try different configurations to find the one that is best suited for their particular project. Please, keep in mind that the images shown in this document may not exactly match the options and distribution shown in the application, since different updates may result in slight changes.