Electromagnetic coupled with Thermal Tutorials for NX-Magnetics

Dr. Binde Ingenieure

January 2, 2024

 Solver Version 850. Valid for Simcenter / NX Versions: 1872.

THIS SIMULATION SOFTWARE USES FINITE ELEMENT METHODS. USERS SHOULD BE AWARE THAT RESULTS CAN HAVE UNPREDICTABLE ERRORS IF INPUT DATA IS NOT COMPLETELY CORRECT. THEREFORE ANY DESIGN DECISIONS SHOULD NOT BE BASED SOLELY ON THE SIMULATION. USE ADDITIONAL MEASUREMENTS TO ENSURE THE CORRECTNESS.

Contents

1 Coupled Thermal (internal)

This chapter shows different ways of electromagnetic heating problems and corresponding simulations. They are all solved using the Magnetics solver and the internally, tightly coupled thermal solver. This thermal solver can handle thermal conduction, convection with fixed coefficients, radiation to environment and some more basic boundary conditions. If additional thermal effects must be captured, there exists the possibility to couple to different solvers like Simcenter Thermal/Flow or ESC, what is shown in later chapters of this guide.

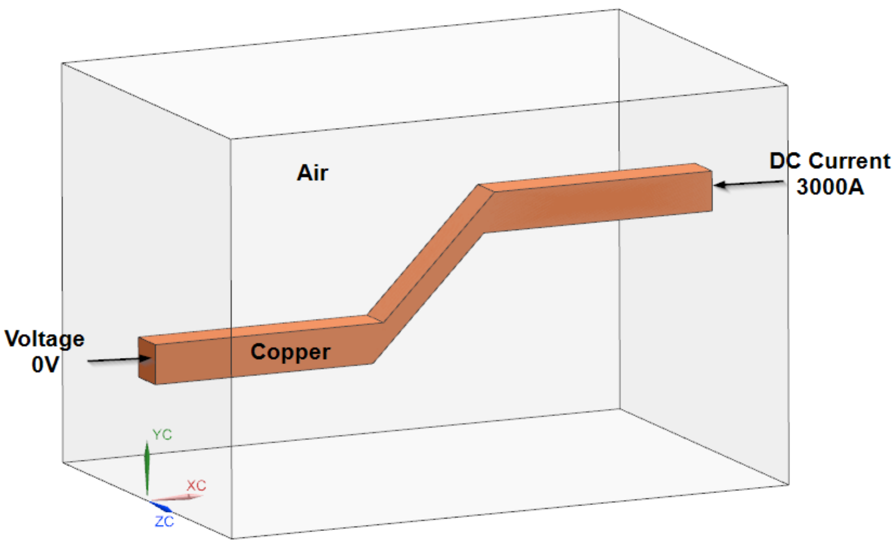
For the conductor body of all following simulations, we use the material 'Copper (Cu-ETP, CW0048)' with properties from 'Deutsches Kupferinstitut, www.kupferinstitut.de'.

1.1 General Principles

Important for understanding the differences between the tutorials is to make clear, how the different solution types are defined. The simulation is divided into two domains. The electromagnetic domain and the thermal domain. Each of them can be defined as transient or steady state. A transient defined thermal domain means a temperature which varies over time. The steady state simulation gives the value of the converged transient temperature. Transfered to the electromagetic domain the load (current or voltage) is defined either transient or steady state. Transient means a changing load over time, steady state means the value of the settled transient load. The current can either be DC or AC.

1.2 Conductor Heating with DC

A conductor is heating up due to DC current of 1500 A. The solution will use a constant current, analyse for the transient electromagnetic field and temperature distribution. The results are compared against simple analytic analysis methods. Some following examples build on top of this one. They analyse for the steady state temperature of the conductor, and for AC conditions.



Estimated time: 1 h

Follow the steps

1.2.1 File Structure and Solutiontype

1. Download the model files for this tutorial from the following link:
   `https://www.magnetics.de/downloads/Tutorials/7.CouplThermal/7.1HeatingConductor.zip`

2. Open the CAD part file 'HeatingConductor.prt'.

3. Start Simcenter Pre/Post, load the CAD part and create a new FEM and Simulation using Solver 'Magnetics' and 'Analysis Type' '3D Electromagnetics'. Switch off the idealized part. For the creation of the solution use the following settings:

4. Name the solution 'HeatingDC',

5. Use the Solution Type 'Magnetodynamic Transient',

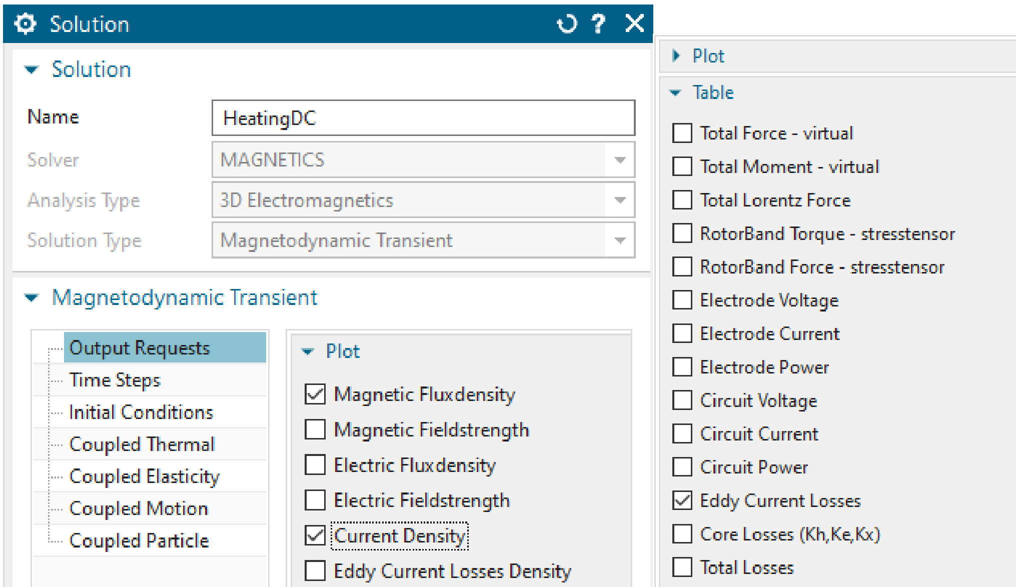
   Hint: This solution type will find the dynamic electromagnetic fields in the conductor and in the air. A simplification would be to use the 'DC Conduction Steady State' solution that would solve only in the conductor.

6. In register 'Output Requests':

   In Plot: Activate 'Current Density',

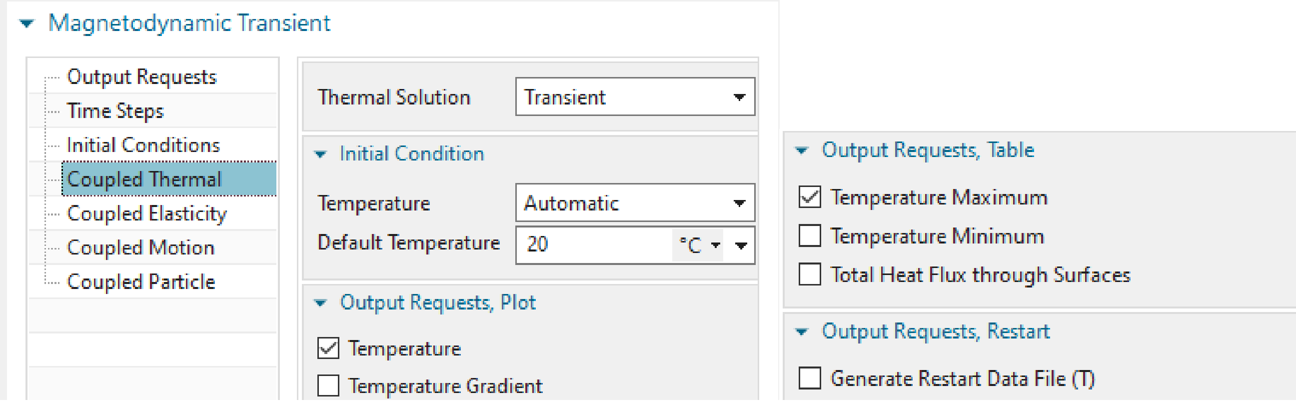
   In Table: 'Eddy Current Losses'.

   Hint: Instead of the Eddy Current Losses, one can also use the 'Total Losses'. The total losses are eddy + core losses and in this example, we will not activate core losses.

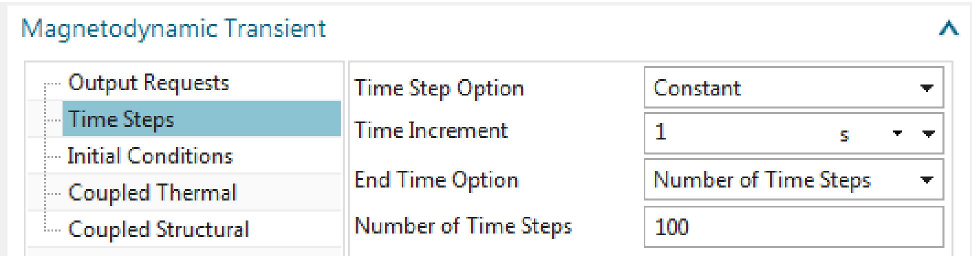


7. In register 'Coupled Thermal': Set the 'Thermal Solution' to 'Transient' and accept the default initial temperature of 20^0C to assign this temperature as default condition to all temperature conducting parts. Check that the output request 'Temperature' is on and

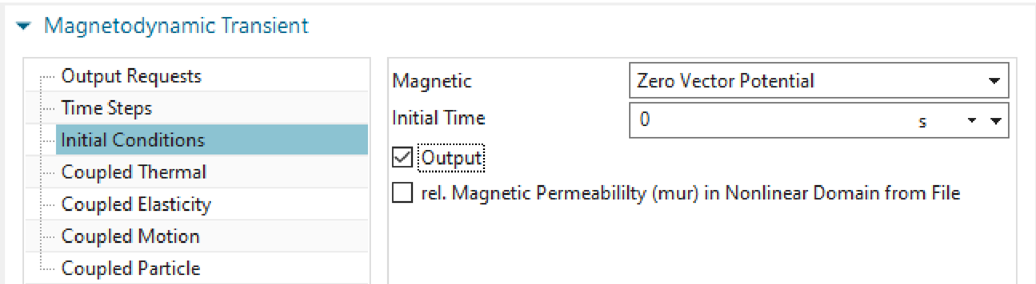
activate 'Temperature Maximum'.



8. In register 'Time Steps': Our solution time will be done in 100 steps, each with an increment of 1 seconds.



9. In register 'Initial Conditions': Activate 'Output' to get the initial time step result written into the result files.



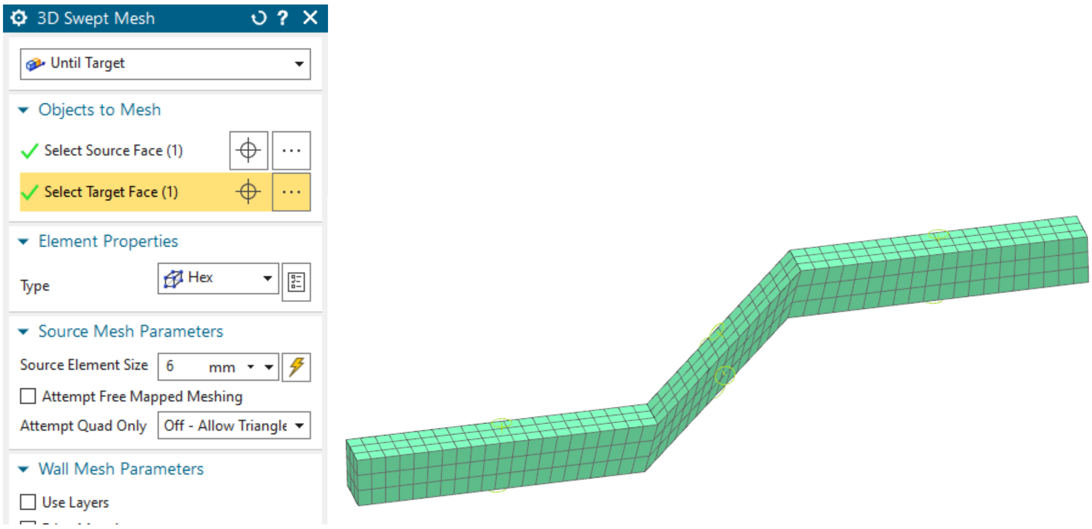
10. Click OK.

1.2.2 Fem File Steps

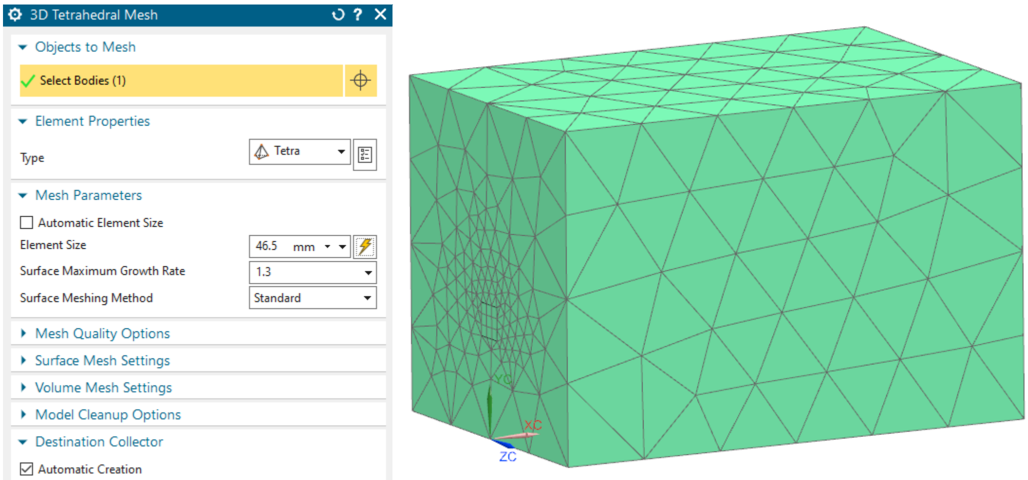
1. Switch to the Fem file

2. If the 'Early Access Feature for Electro Magnetics' is not 'On' (see chapter 'Recommended Settings'), Mesh-Mating-Conditions (MMC) over all geometry now have to be created. If the feature is 'On', MMCs are not necessary. In that case it only may be checked that the group 'non-manifold face' shows the correct adjacent faces.

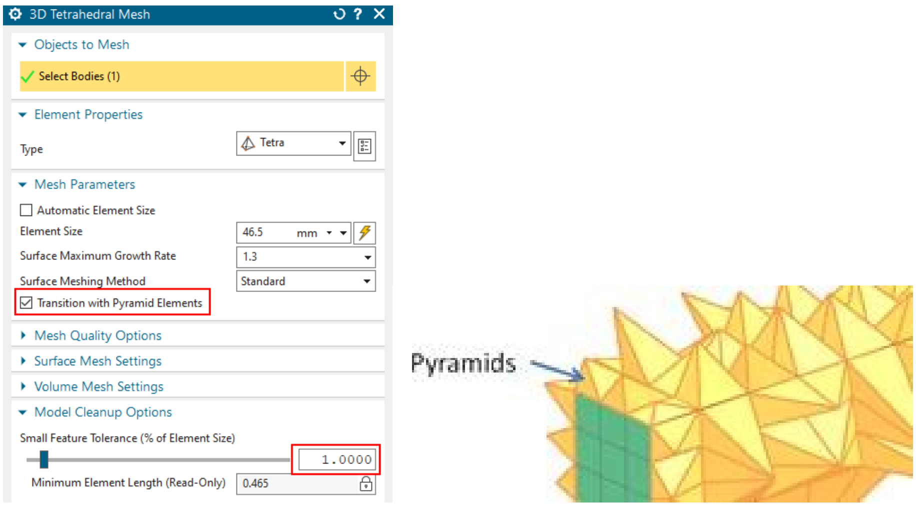
3. Mesh the conductor using hex elements (tets are also possible). Choose an appropriate element size (i.e. 6 mm).



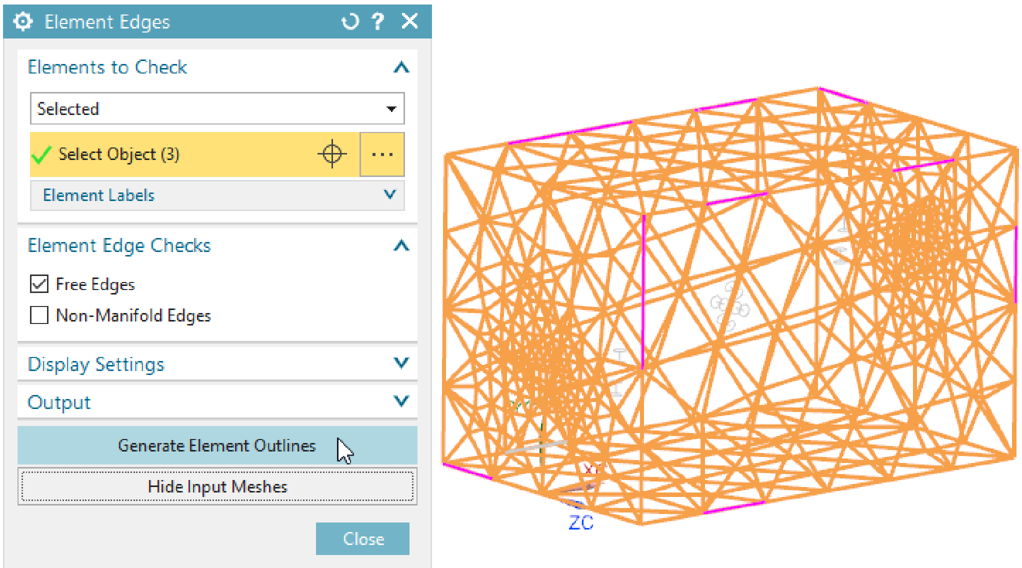
4. Mesh the air volume with tets.



5. If MMCs are used, the creation of pyramids for perfect transition is possible as seen in the below picture.



6. Maybe you do a check 'Element Edges' and verify there are no free edges inside the whole region.

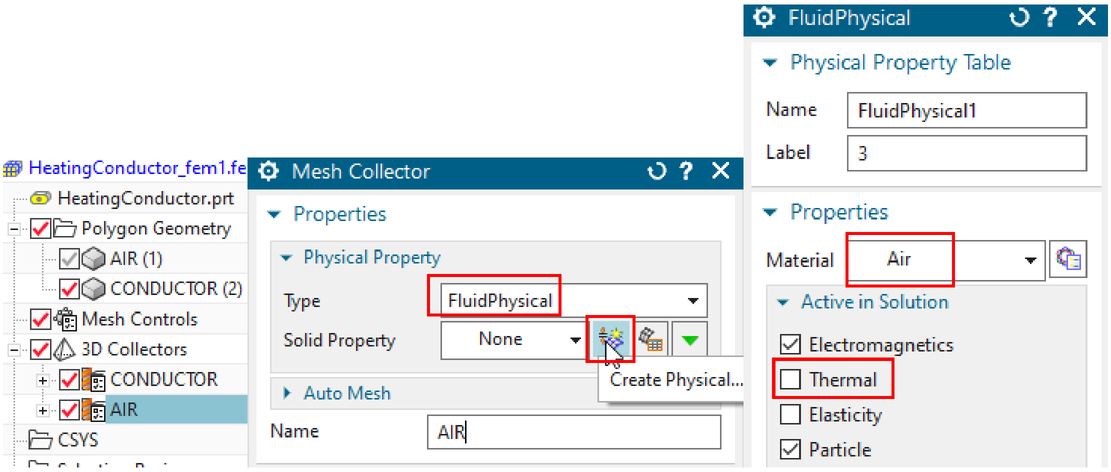


1.2.3 Materials and Physical Properties

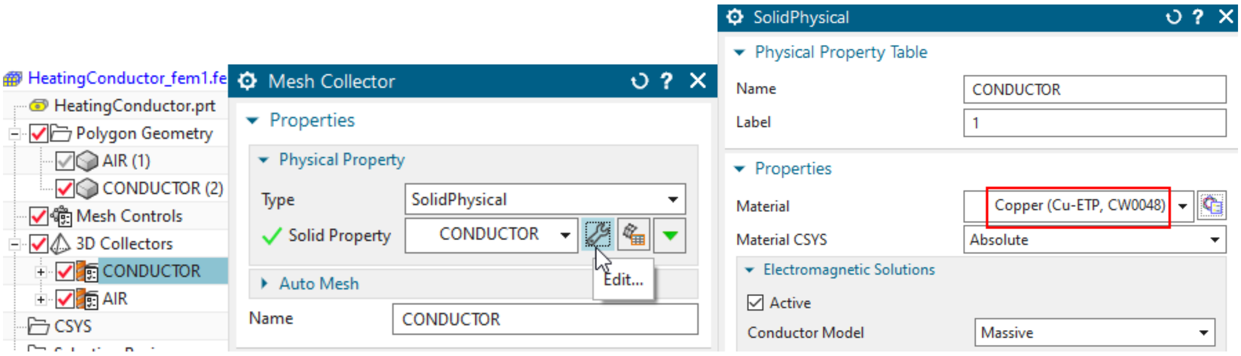
1. For the 'Air' mesh-collector, use a physical of type 'FluidPhysical' and choose the material 'Air' from the Magnetics material library.

2. At 'Active in Solution', check that 'Thermal' is deactivated (default). The conductor will not be solved thermally.

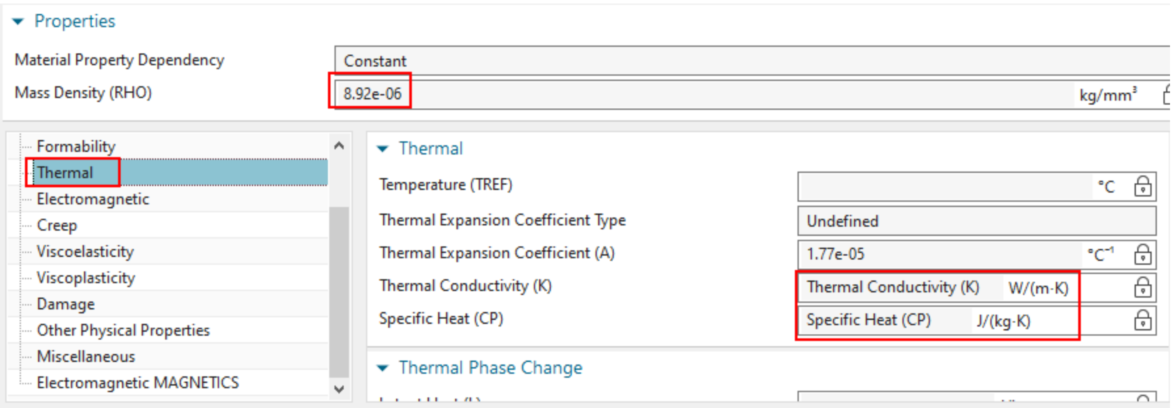
   Hint: This is necessary, because we later will apply thermal convection on the conductor outside walls. Thus, these convection coefficients will describe the thermal loss of the conductor. If the air would be additionally active, we would additionally simulate thermal loss from the thermal conduction between the copper and air.



3. Assign material the 'Copper (Cu-ETP CW0048)' from the magnetic library to the conductor. The important material properties are explained in the following (To check them, use the 'Inspect' button on the material).

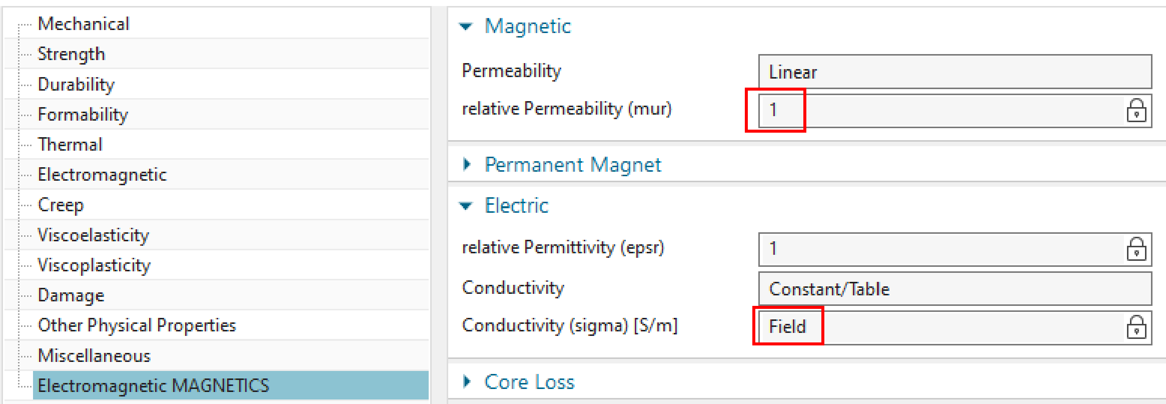


In register 'Thermal', all properties for the thermal simulation are defined. These are:



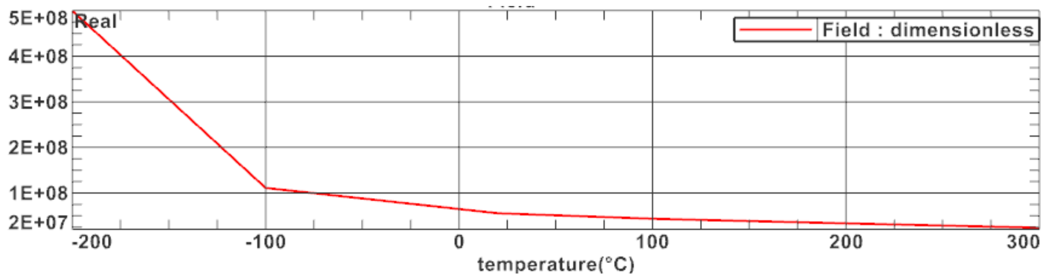
- Material 'Mass Density (RHO)' together with
- 'Specific Heat (CP)'. These both are necessary to model transient thermal effects.
- The 'Thermal Conductivity (K)' is needed for thermal conduction.

In register 'Electromagnetic MAGNETICS' the properties for the magnetic problem are defined as follows.



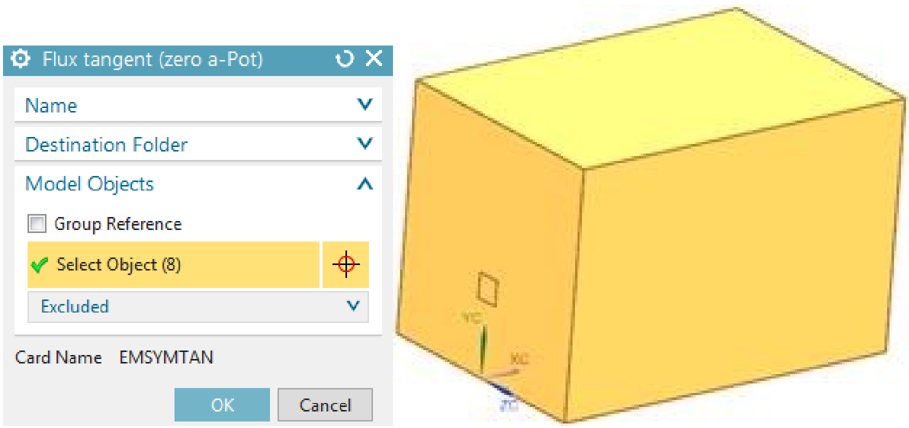
- In box 'Magnetic': The 'relative Permeability (mur)',
- In box 'Permanent Magnet': The 'Remanent Magnetic Fieldstrength X' in case of permanent magnets and
- in box 'Electric': The electric 'Conductivity (sigma) [S/m]'.

- Several properties are defined temperature dependent, as seen in the below picture for electric conductivity [S/m]. Tables are recognized by a text-name in the corresponding field instead of a numeric value.

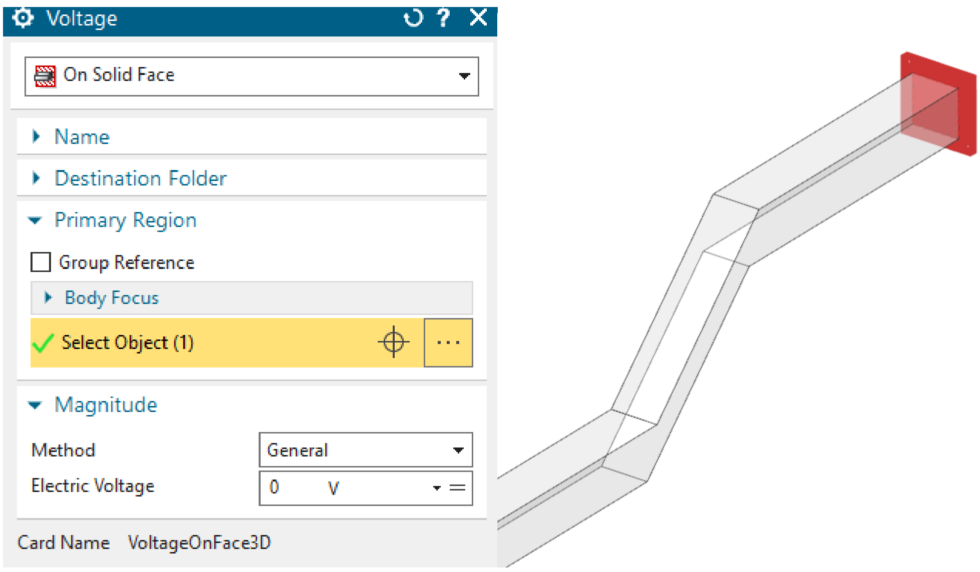


1.2.4 Sim File Steps

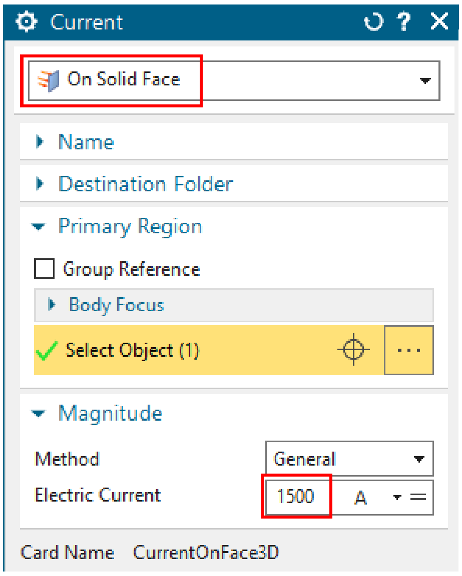
1. Switch to the Sim file, blank all meshes.
2. Create loads and constraints for the usual electromagnetic part of the problem as follows.
   - Create a constraint of type 'Flux tangent (zero a-Pot)' to all 8 outside faces of the air-volume. Assign this condition also to the two electrode faces of the conductor.

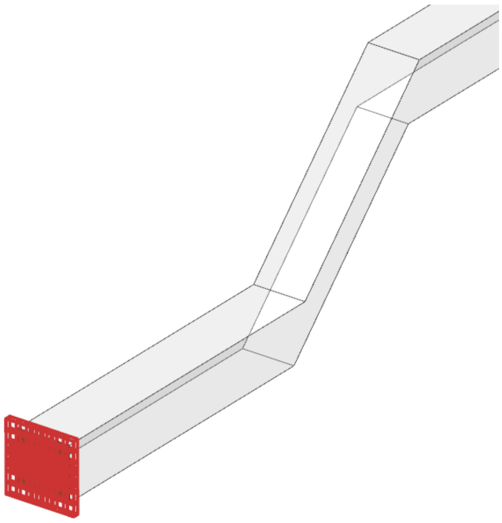


   - Assign a load 'Voltage' of type 'On Solid Face' to one of the electrode faces of the conductor. Assign a value of '0 V'.

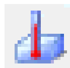


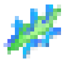
- Assign a load 'Current' of type 'On Solid Face' to the other electrode face of the conductor. Assign a value of '1500 A'.



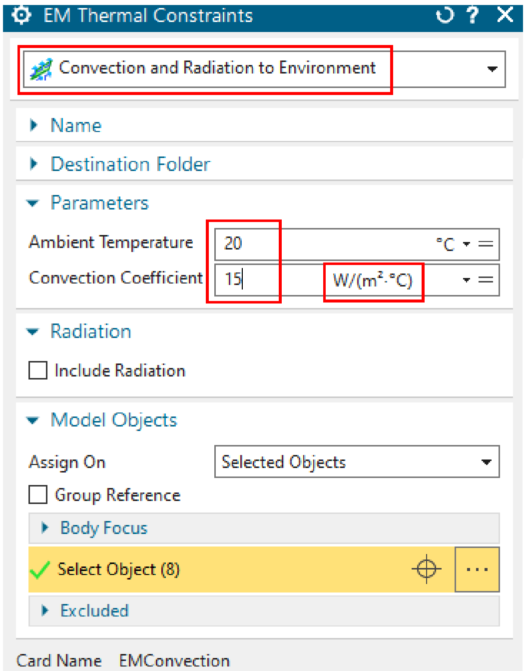


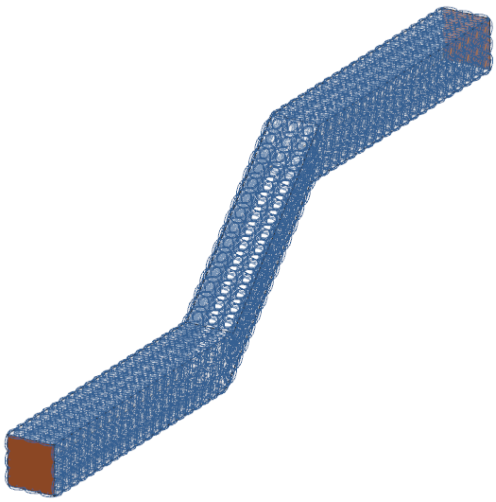
3. Create a convection constraint for the coupled thermal solution:

   - Blank the Air body and choose a constraint of type 'EM Thermal Constraints' .

   - In the dialog set the type to 'Convection and Radiation to Environment' .

   - Select the outside faces of the conductor (all faces except the 2 electrode faces. The number of faces is 8). Assign the values for 'Ambient Temperature' and 'Convection Coefficient' as shown. Take care using the shown units.





4. Solve the solution. The 100 time steps take approximately 2 minutes solve time.

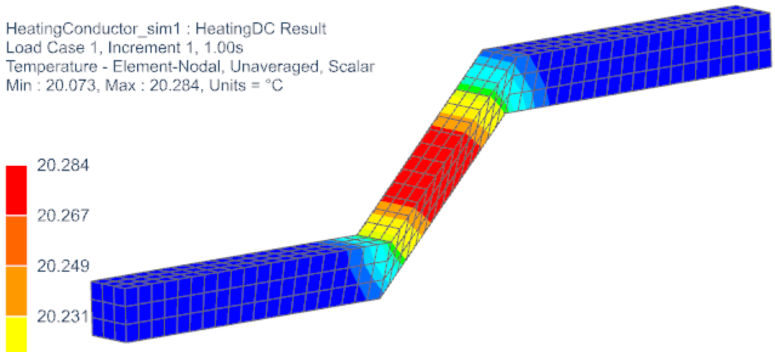
1.2.5 Post Processing

1. Open the plot results and check them as follows.

2. Verify that in the 'Initial Time Condition, 0 sec', there is a temperature of 20°C in the conductor and a current density of zero. This result represents the initial condition and is only of interest for checking.

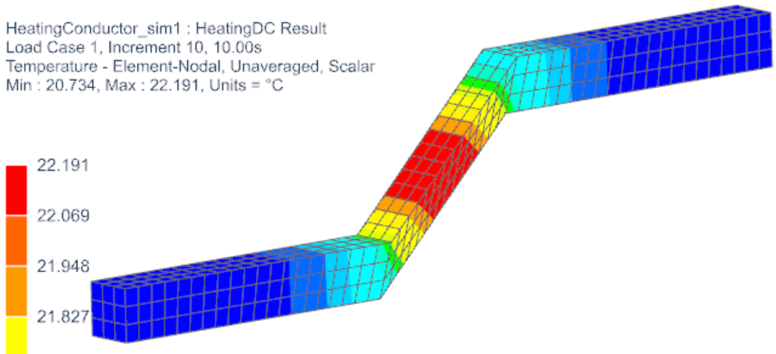
3. Check the result at time 1 sec.

   Temperature is 20.3^0C. Current density in average is about $4.6e6A/m^2$.



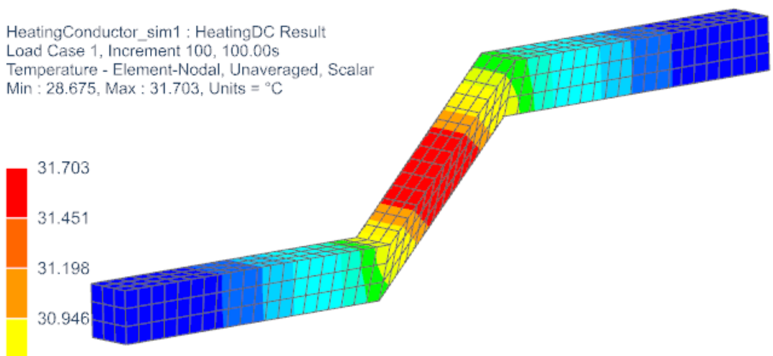
4. Check the result at time 10 sec.

   Temperature in conductor is 22.2^0C.



5. Check the result at time 100 sec.

   Temperature is 31.7^0C.



1.2.6 Verification

We compare against analytic thermal theory using a basic energy form for heating of conductors. Notice that for simplicity reasons this formula does not include convection effects.

$$Q = m \cdot C \cdot \triangle T \;\Rightarrow\; \triangle T = \frac{Q}{m \cdot C}$$

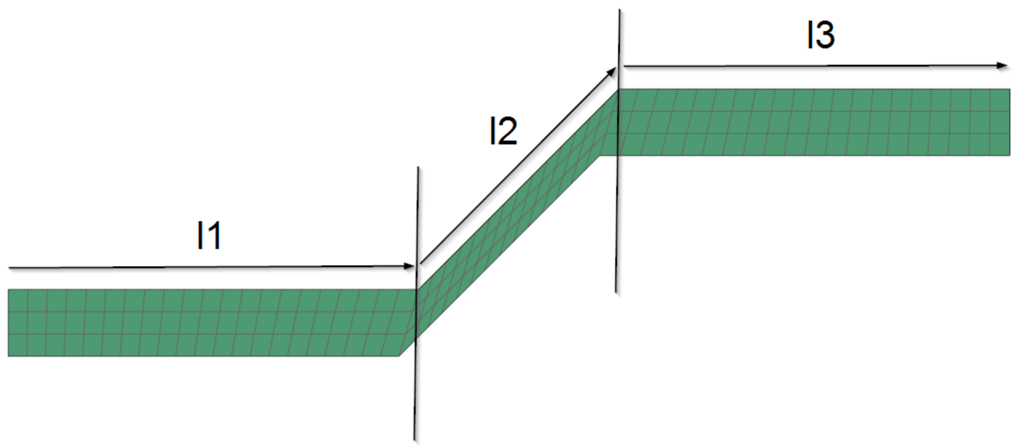
with T: Temperature [K], Q: Energy [W s], m: mass[Kg], C: Capacity [W s/K]. Q can easily be determined by the following equations:

$$Q = P \cdot t$$
$$P = R \cdot I^2$$

with P: Power Loss [W], I: Current[A], t: Time[s], R: specific resistance[$\Omega \cdot \frac{l}{A}$]. The specific resistance R is defined by the reciprocal value of the conductivity multiplied by the the length of the conductor divided by its cross-section. As the conductor exists of different cross sections, the specific resistance of the different areas can be added up:

$$R = \frac{1}{\sigma} \cdot \sum \frac{l}{A}$$

The conductivity $\sigma[S/m]$ can be read from the material property in NX, the length[m] and cross section[m^2] of each area can be measured.

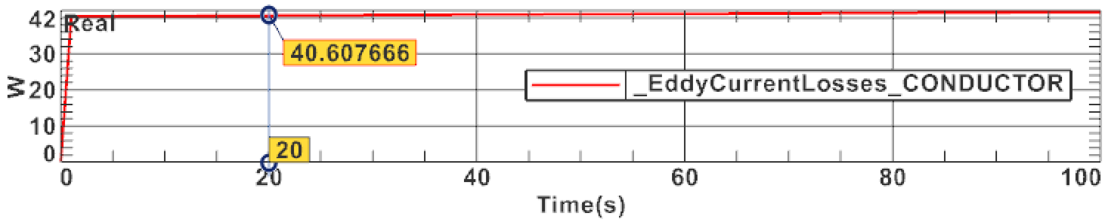


By using an excel sheet the resulting delta T values become:

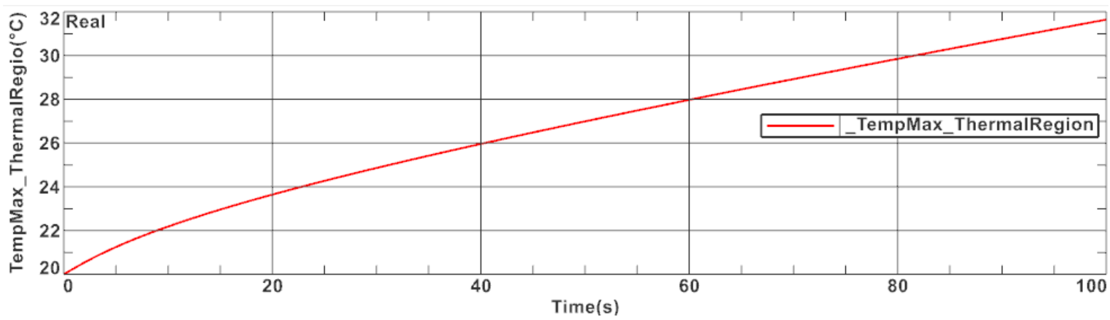
Delta T [sec]	Temperature Theory [°C]	Temperature Simulation [°C]
1	0.1	0.3
10	1	2.2
20	2.1	3.7
40	4.2	6.0
100	10.5	11.7

which is acceptable close to our simulation results. (subtract 20°C from the simulation results because of the initial temperature). At larger time periods the result differs more and more because of convection effects included in the simulation.

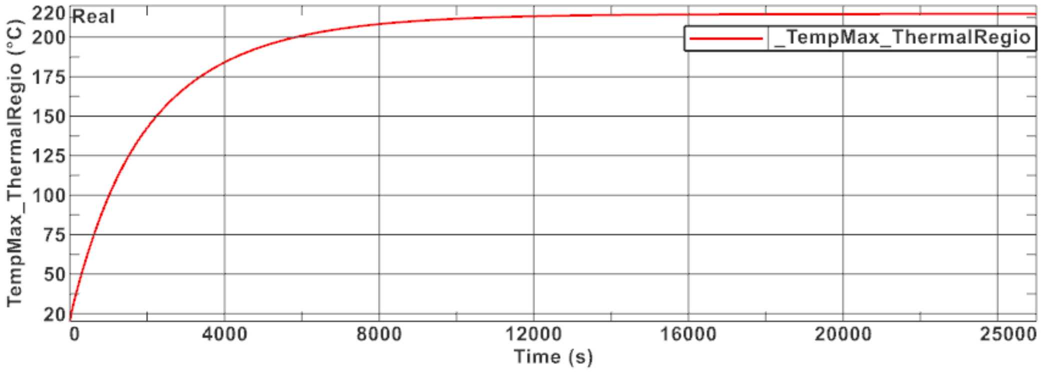
1. Open the afu file that shows eddy current losses. Check the calculated eddy losses:
   - Conductor: about 40.6 W.
   - Using the excel sheet you can check the analytically calculated eddy losses on the conductor. This value is 40.7 W, thus, the simulation losses results are accurate.



2. Display the graph of temperature maximum over the iterations. The graph shows that after 100 s there is still no steady state situation.



3. To optionally simulate a longer time, period perform these steps:
   - clone the solution,
   - Name the new one 'HeatingDCLongTime'.
   - Change the 'Number of Time Steps' to 500 and
   - The 'Time increment' to 50 sec.
   - Solve the solution. This will require about 12 minutes time.
   - Create again the temperature graph over iterations. At the end time the temperature doesn't change very much any more. Thus, the end temperature (214.7^0C) is now near to a steady state situation. We will compare this value in the next example against a simulation that directly solves for steady state.
   - The eddy current losses have increased to 66.9 W. The reason is that at higher temperatures the electric conductivity decreases and if the current is constant, the losses must increase. We will compare this also against the following steady state solution.



4. Save your files. In case you want to proceed with the next example, don't close the files.

1.3 Conductor Steady-State Temperature from DC

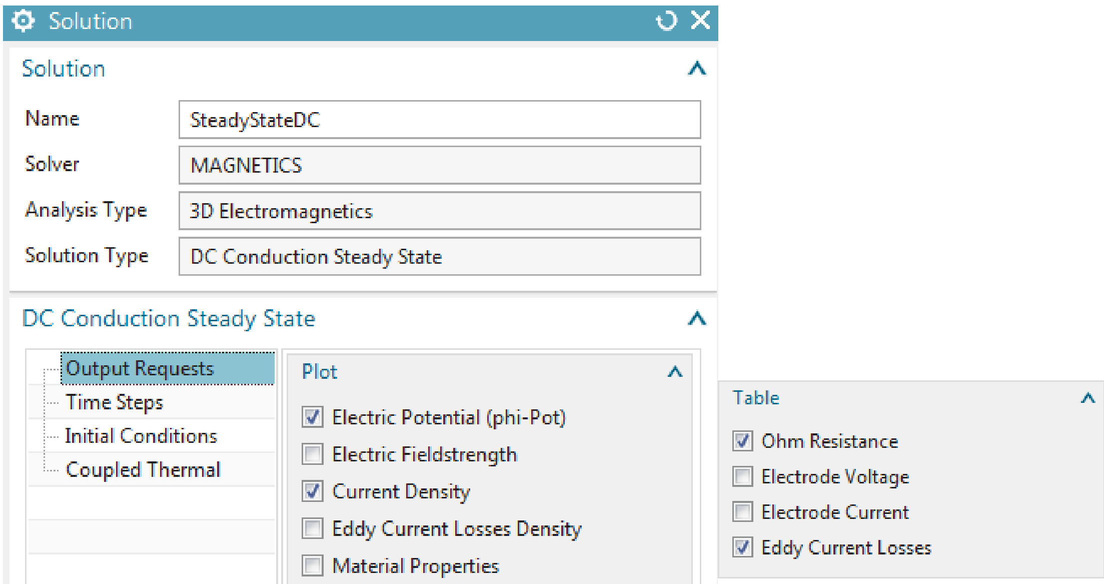
This example builds on top of the last one and analyses for the steady state temperature of the conductor.

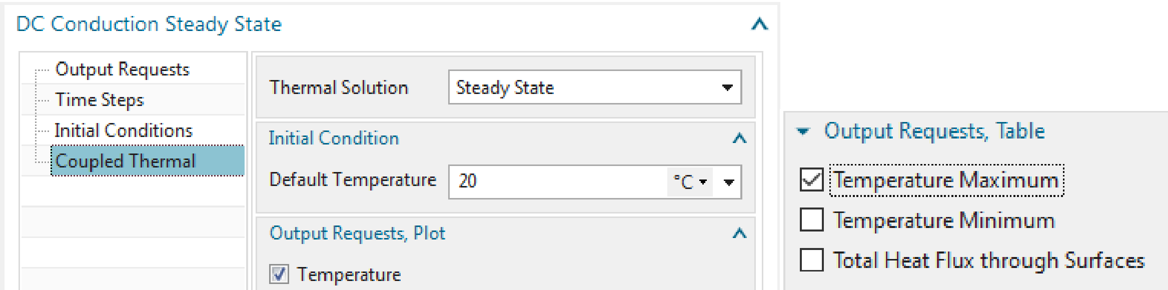
Estimated time: 10 min.

Follow the steps to reproduce it:

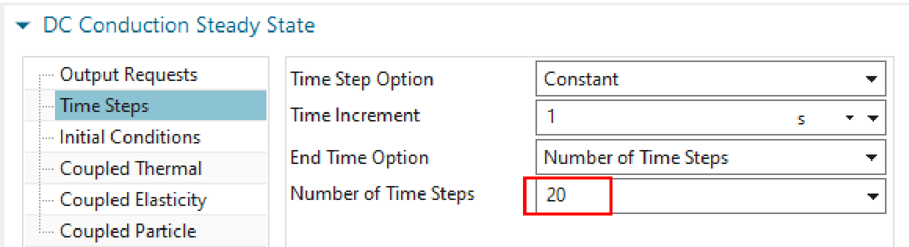
1. Open the Sim file 'HeatingConductor_sim1.sim' generated by the last example. You can either use your own files or those from the 'complete' folder.

2. Create a new solution of type 'DC Conduction Steady State'.
   Name this solution 'SteadyStateDC'. Activate the output requests and thermal solution as shown below.



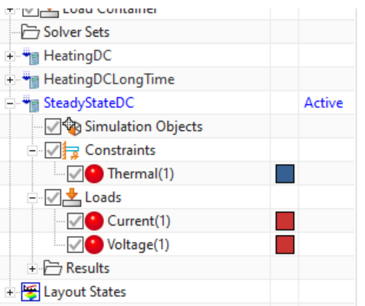


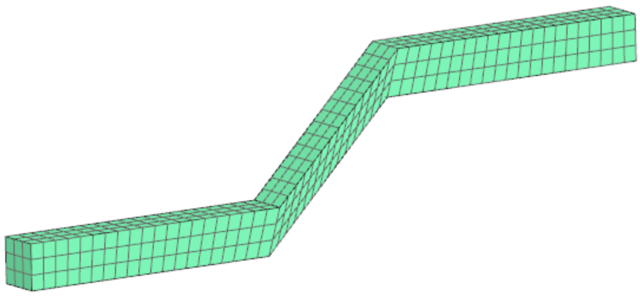
3. At Time Steps, we should allow some steps because of temperature dependent material properties: These properties are updated with each time step and thus the solution becomes more and more accurate. Let's use 20 steps and in the result we will see that already 5 steps would be enough.



4. Re-use the 'Thermal' Constraint with convection. You can do this by 'drag and drop' the constraint from the container into the constraints of the new solution.

5. Re-use the 'Current' and 'Voltage' loads in the same way.



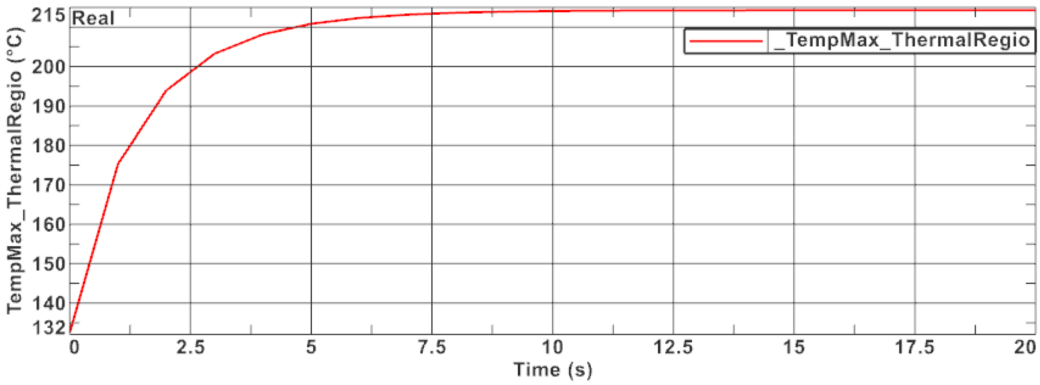


6. Solve the solution.

   Hint: The air mesh doesn't play a role in this simulation. Therefore, it can be deleted, but it can also stay. Another way would be to set it into a simulation object 'Deactivation Set'.

7. Open the results and check them:

   - The Current Density result in the middle of the conductor is about $7.2e6 A/m^2$. This corresponds to the loaded value of 1500 A on the face area of $20mm \cdot 10mm$.
   - The temperature curve starts at $132.5^0 C$ and increases up to 214.7 what is the same value as the long time transient solution gave us. After about 4 steps there is nearly no change any more. The curve represents material properties as they update with temperature changes.



- Open the corresponding graph file. Check the computed loss value on the conductor after some (4) steps is again 66.5 W as already in the transient solution after long time.
- Check the computed ohm resistance. It starts at 1.79e-5 Ohm and finishes with 3.09e-5 Ohm. The start value is near to the analytically value of 1.81e-5 Ohm because the analytic form did not take into account the temperature dependency of material properties.

8. Save your files. If you want to proceed with the next example don't close them.

1.4 Conductor Heating by AC, Steady State

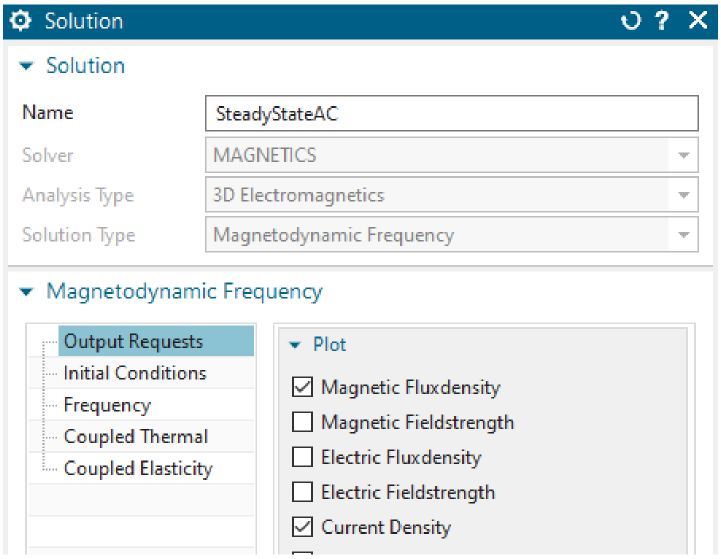
In this example we will apply harmonic current (AC) on the conductor geometry. In the result we will find eddy current losses on the conductor. Those losses will lead to heating of the conductor.

Estimated Time: 15 min.

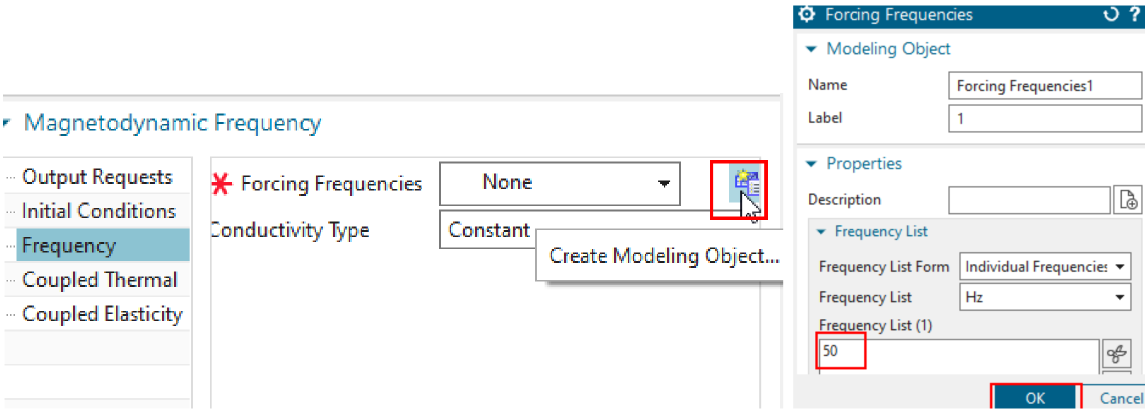
In this first part we will do a magneto-dynamic frequency solution for the AC magnetic fields with a coupled steady state solution for the thermal field. This will result in the steady state solution for the temperature under AC condition.

Follow the steps:

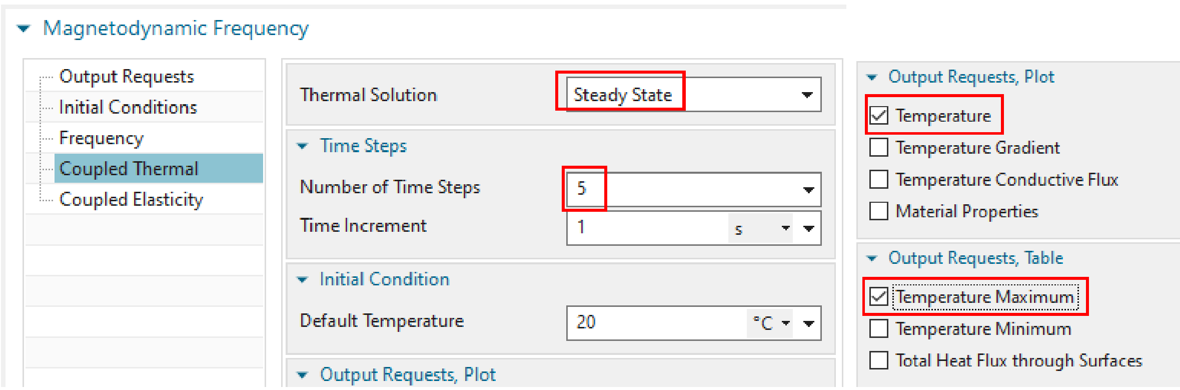
1. Open the Sim file 'HeatingConductor_sim1.sim' generated by the last example. You can either use your own files or those from the 'complete' folder.
2. Create a new solution of type '3D Magnetodynamic Frequency'.
   Name this solution 'SteadyStateAC'. Activate the settings as shown:



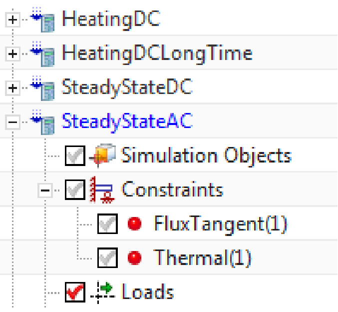
3. In register 'Frequency', create a modeling object for the forcing frequency and accept the default 50 Hz in the list.



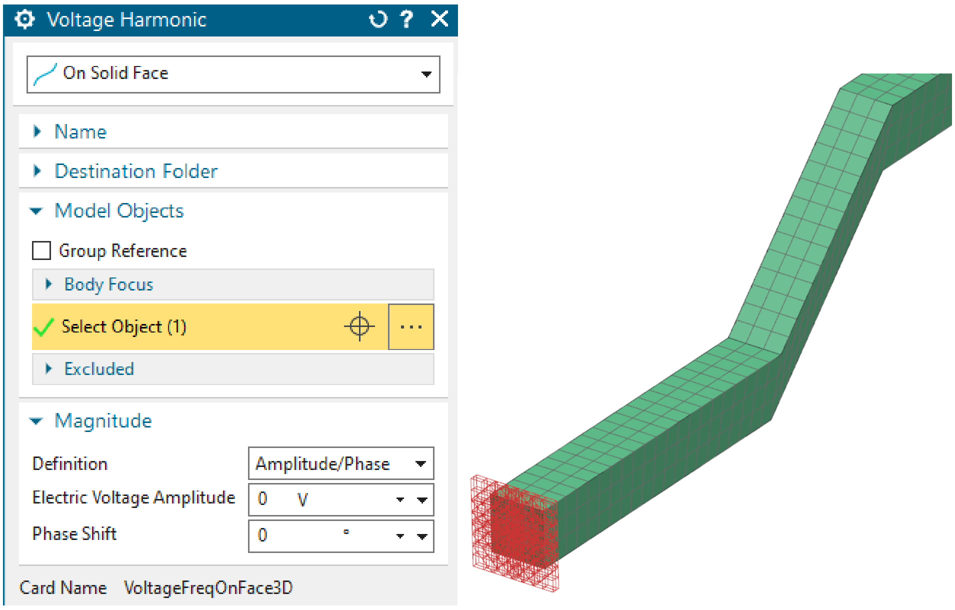
4. In register 'Coupled Thermal', set the 'Thermal Solution' to 'Steady State'. To allow temperature dependent material properties to converge, we set the number of time steps to 5. The resulting temperature curve will show the convergence behaviour.



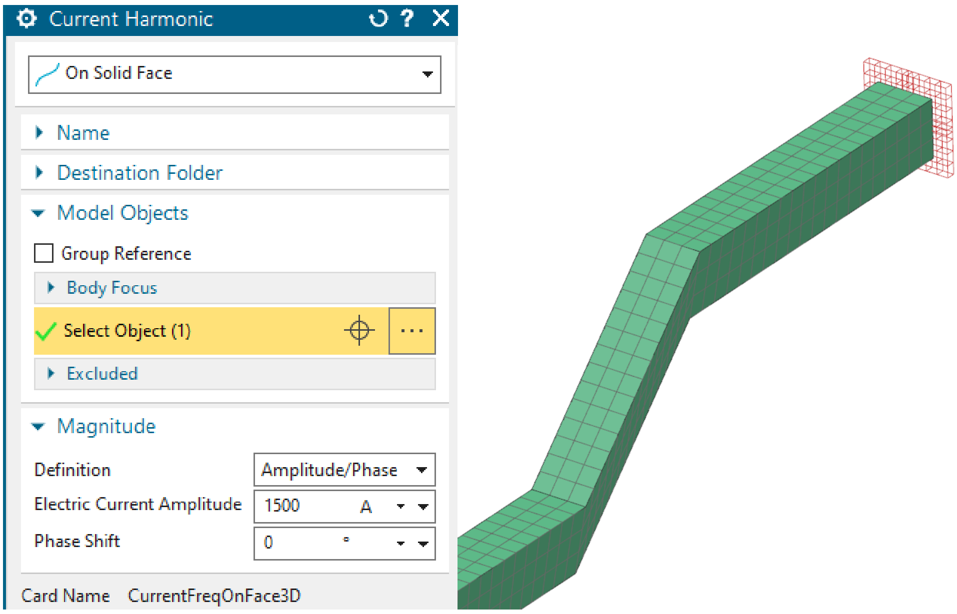
5. Use both already created constraints 'FluxTangent', and 'Thermal' for this solution.



6. Create a load 'Voltage Harmonic', set the type to 'On Solid Face' on one of the electrode faces. Assign 0 V.

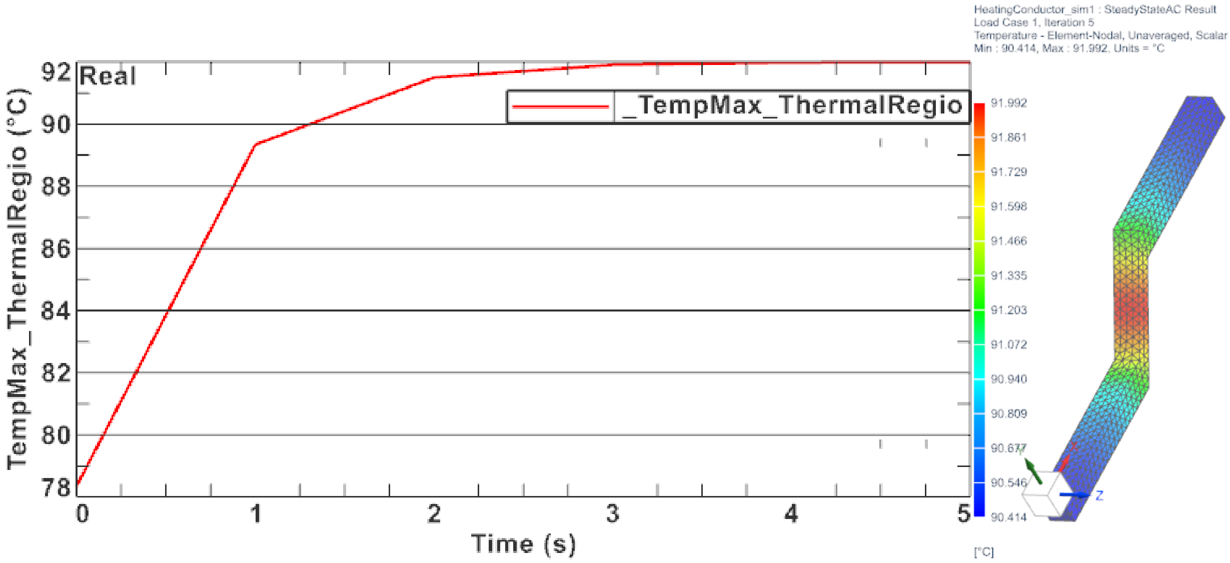


7. Create a load 'Current Harmonic' on the other electrode face. Set the type to 'On Solid Face'. Assign 1500 A.



8. Solve the solution.

9. Open the results and display the temperature graph and the plot results. The converged temperature of about 91.99^0C resulting from this AC analysis is lower than the one from DC analysis even if both use 1500 A. The reason is the lower effective power in AC.



10. Save your files and close them.

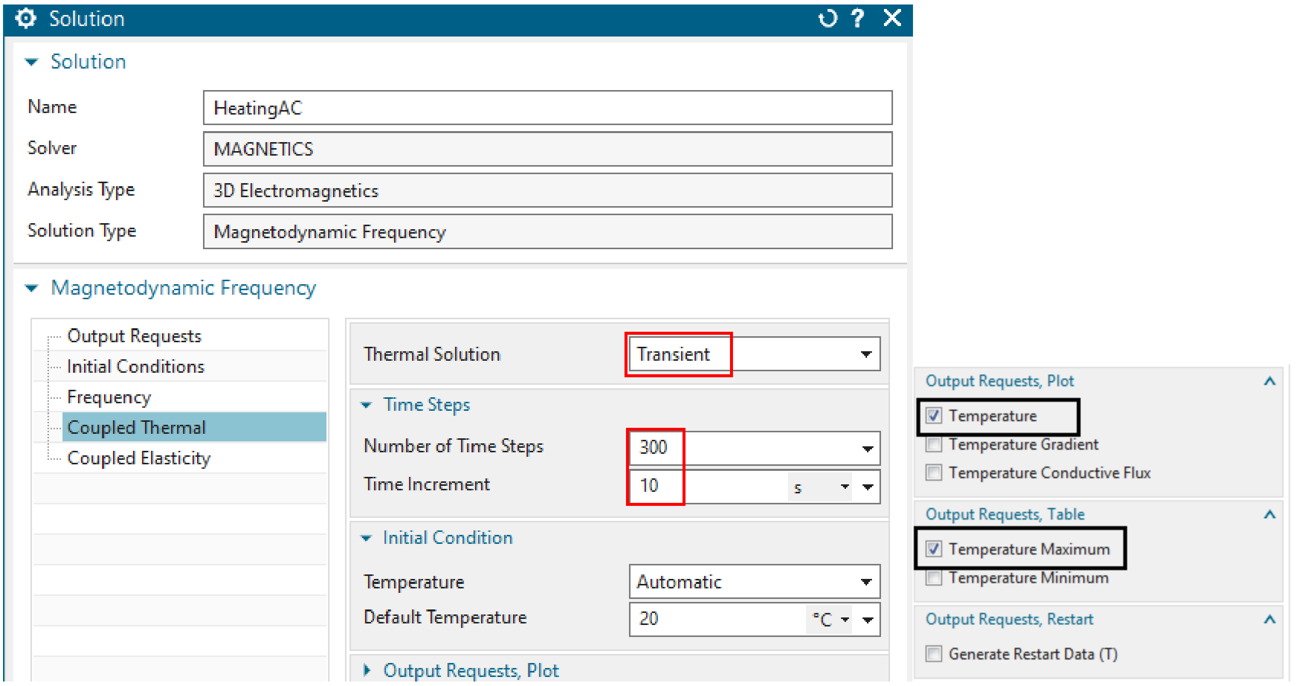
1.5 Conductor Heating with AC, Transient

We do the same as in the last example, except the thermal domain will be set to transient. This will result in a step by step heating of the conductor, starting from the initial temperature of 20^0C. When doing this over a long time period, result will be the same as in the steady state case of the last example.

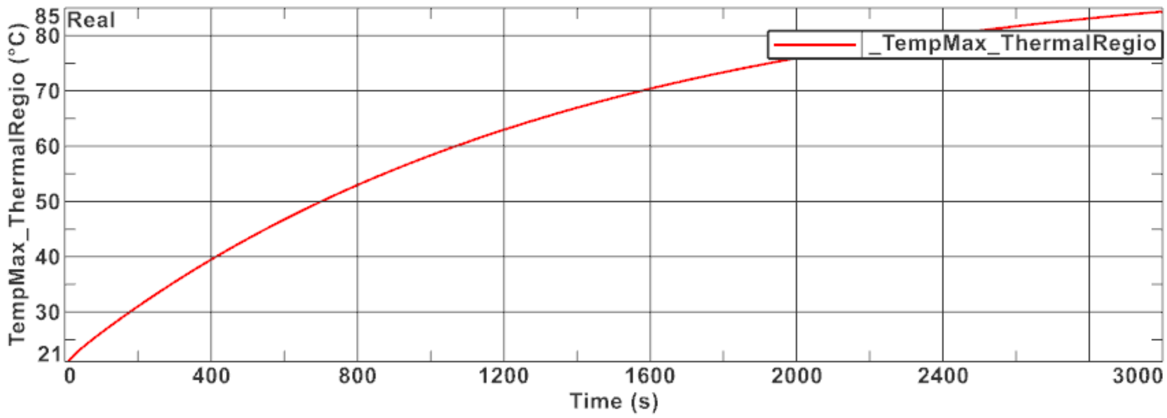
1. Open the Sim file 'HeatingConductor_sim1.sim'.

2. Clone the last created solution 'SteadyStateAC'. Rename the new to 'HeatingAC'.

3. Edit the solution parameters. In register 'Coupled Thermal' set the 'Thermal Solution' to 'Transient' and set the settings as shown below.



4. Solve the solution.
5. Open the results and show the graph for the conductor maximum temperature. The temperature at the end of the time period becomes 84.3^0C and it corresponds adequate to the one calculated in the previous steady state case (91.99^0C). Deviation: 9.1%

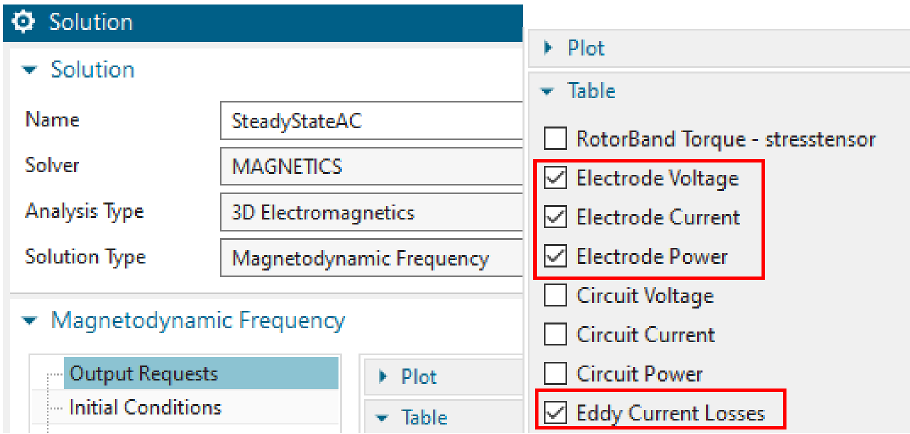


6. Save your parts and close them.

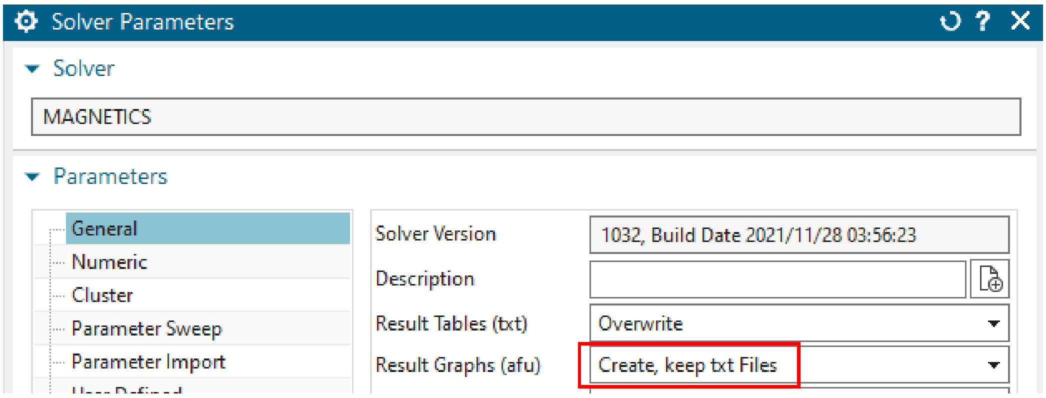
1.6 Understanding Sources and Losses in Frequency Solutions

At each electrode there is either voltage (U) given and current (I) will result or I is given and then U will result. If the solution runs in frequency domain, such as 'Magnetodynamic Frequency', then all results are complex, meaning they have an real (Re) and an imaginary (Im) part. For the following explanations we use solution 'SteadyStateAC'.

- RMB 'Edit' the solution and activate the following table output requests: 'Electrode Current', 'Electrode Voltage', 'Electrode Power' and 'Eddy Current Losses'.



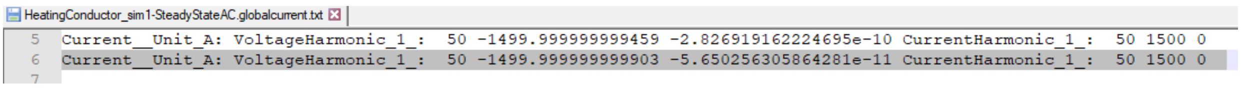
- RMB 'Edit Solver Parameters' for the solution and set the option 'Result Graph Files' to 'Create, keep txt Files'. This setting allows us to find the tabular results in simple text files.



- Solve the solution.

- After solve has finished, check for files with 'txt' extension in the work directory. There should be one txt file for each requested table result. Such txt files show in one line all results of the electrodes always in this manner: First value is the frequency, second the real value (Re) and third the imaginary value (Im). Using these values many others can be derived, for instance the amplitude is the geometric addition as $\sqrt{Re^2 + Im^2}$

- Lets start with the current results. Open the file with extension '.current.txt' and check the last line (each line is for one time step). It can be seen that the electrode named 'VoltageHarmonic 1' has a current of Re = -1500 A (nearly) and Im = 0 A (nearly). The second electrode 'CurrentHarmonic 1' has Re = +1500 A and Im = 0 A. This second result is not very interesting, because it is what we have assigned, but the first one is a result from the simulation and it shows great accuracy.


HeatingConductor_sim1-SteadyStateAC.globalcurrent.txt

```
5  Current__Unit_A: VoltageHarmonic_1_:  50 -1499.999999999459 -2.826919162224695e-10 CurrentHarmonic_1_:  50 1500 0
6  Current__Unit_A: VoltageHarmonic_1_:  50 -1499.999999999903 -5.650256305864281e-11 CurrentHarmonic_1_:  50 1500 0
7
```

- Now lets check the resulting voltages: Open the file with extension '.voltage.txt'. We see that the electrode 'VoltageHarmonic 1' has a voltage of Re = 0 V and Im = 0 V. This is what we have assigned. The second one 'CurrentHarmonic 1' has Re = -0.03059 V and Im = -0.07700 V. This represents the voltage drop over the conductor.

HeatingConductor_sim1-SteadyStateAC.globalvoltage.txt

```
Voltage__Unit_V: VoltageHarmonic_1_:  50 0 0 CurrentHarmonic_1_:  50 -0.03059460342803641 -0.07700206512190756
Voltage__Unit_V: VoltageHarmonic_1_:  50 0 0 CurrentHarmonic_1_:  50 -0.03059482909482241 -0.07700206687558346
```

- Now check the electrode power results. Open the file with extension '.ElectrodePower.txt'. The following results are computed from the prior U and I by complex multiplication. So we find Re = 0, Im = 0 for 'VoltageHarmonic 1' and Re = -22.94612 W, Im = -57.75155 W for 'CurrentHarmonic 1'. These give important information: The Re value is the active or effective power (german Wirkleistung) and the Im value is the reactive power (german Blindleistung).

  HeatingConductor_sim1-SteadyStateAC.ElectrodePower.txt

  ```
  Power__Unit_W: VoltageHarmonic_1_:  50 0 0 CurrentHarmonic_1_:  50 -22.94595257102731 -57.75154884143067
  Power__Unit_W: VoltageHarmonic_1_:  50 0 0 CurrentHarmonic_1_:  50 -22.94612182111681 -57.75155015668759
  ```

  In other situations, we may have several electrodes on a conductor, each showing different values for power. Summing up all the effective powers (with correct sign) will give the total effective power loss on that conductor.

- Finally lets check the eddy losses file with extension '.EddyCurrentLoss.txt'. This file now gives not an electrode but an integral result over the whole conductor body. It is calculated by the form $sigma * SquNorm[-Dt[a] - dv]$ what corresponds to the simple form $R \cdot I^2$. The value computed on the conductor is 22.87114 W and it well corresponds to the prior effective power.

  HeatingConductor_sim1-SteadyStateAC.EddyCurrentLoss.txt

  ```
  EddyCurrentLosses__Unit_W: CONDUCTOR:  50 22.87081284134689 0
  EddyCurrentLosses__Unit_W: CONDUCTOR:  50 22.87114413494924 0
  ```

The tutorial is complete. Save your files and close them.

2 Coupled Thermal (external, SC Thermal/Flow/ESC)

This chapter again deals with heating of conductors. Additionally to the chapter before, thermal solutions are now performed by the solver(s) Simcenter Thermal/Flow or ESC (Electronic Systems Cooling). Main advantages of these processes come from the flow cooling effect that is now captured precisely with CFD simulation instead of using simple fixed convection coefficients as we did before.

The Magnetics solver again computes the electromagnetic fields with all corresponding skin- and proximity effects and eddy-current-losses. If the electric current is only DC, the Magnetics solution would even not be necessary, because this kind of pure electric load can be modeled completely in Simcenter Thermal/Flow(ESC) by the feature 'Joule Heating'. As soon as AC or transient currents play a role, this can not be modeled by 'Joule Heating' any more, because of induction effects that must be simulated by dynamic electro-magnetic solvers.

Focus in the tutorials will be on the transfer of such losses from Magnetics to Thermal. But also material properties being temperature dependent play a role. The reader should already be familiar with the solver Thermal/Flow or ESC.

2.1 Conductor Heating with Flow Cooling

2.1.1 One Way Coupling

In this tutorial we start with a quite simple process: First computing the eddy current loss by the Magnetics solver. The result is a single integral value in unit watt (W). Then, using the Thermal/Flow(ESC) solver, we assign that loss value as a load and compute temperatures.

The result of such a simulation is of acceptable quality, it includes the dynamic electromagnetic effects, but it does not capture the spatial distribution of losses. Thus, if more accuracy is required, we can use a loss field instead of the integral value. Below, we show both ways, marked as 'Alternative 1' and 'Alternative 2'.

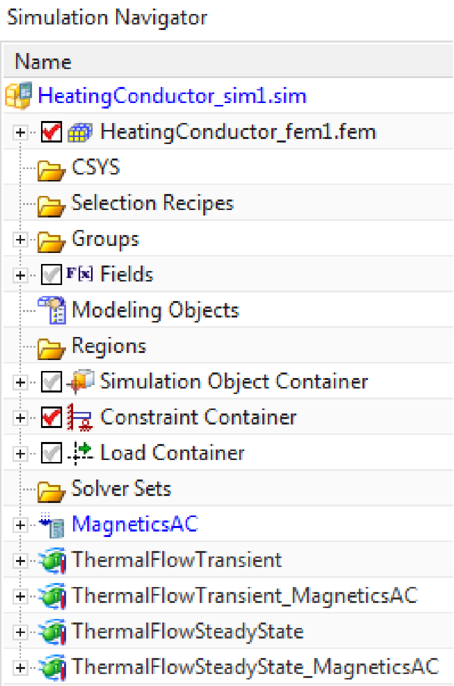
Secondly, not taken into account with such simulations are temperature changes that lead to changes of material properties (mainly the electric conductivity). Thus, this is a simplified process and this limitation is carried out in following tutorials.

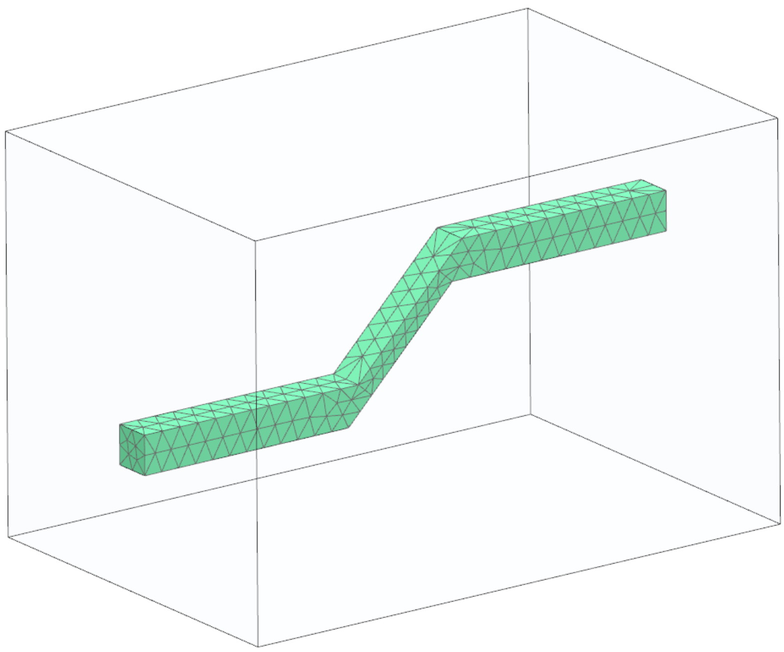
Hint: The model is already build and ready to solve. In this tutorial we go through the existing features to check and explain them.

Follow the steps:

1. Download the model files for this tutorial from the following link:
   `https://www.magnetics.de/downloads/Tutorials/7.CouplThermal/7.2HeatingCond_CoupledFlow.zip`

2. Start Simcenter and open the Sim file 'HeatingConductor_sim1.sim'.





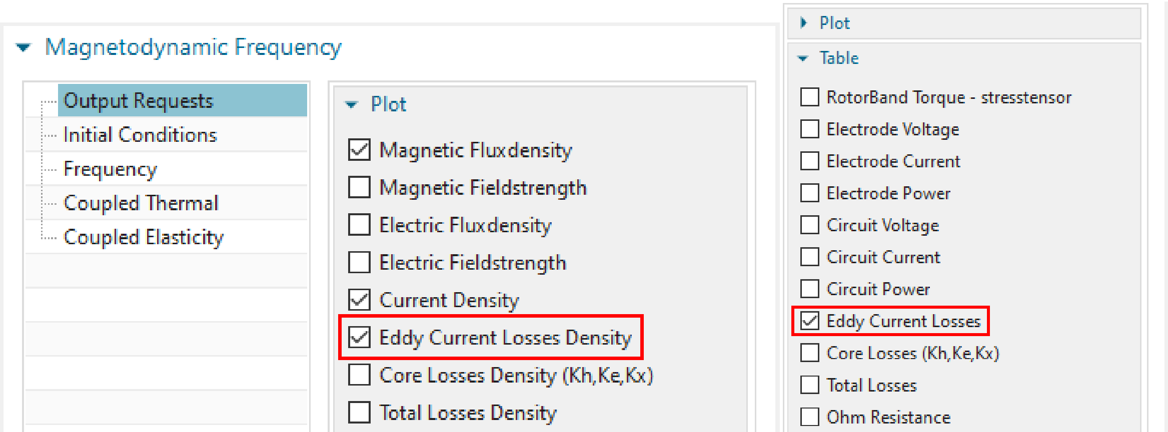
3. Check solution 'MagneticsAC'

   - Edit the solution. This is a Magnetics solution of type 'Magnetodynamic Frequency'. So, it is possible to compute AC with eddy current losses here.

     Hint: This solution and model is quite similar to the AC simulation of the previous chapter. Only, the air mesh uses boundary layer elements for more accurate wall conditions.

- Check the settings in register 'Output Requests':
- The option 'Eddy Current Losses' in box 'Table' is the important output because it computes the integral value of losses. These losses are assigned as load in the Thermal/Flow or ESC solution.
- The option 'Eddy Current Losses Density' in box 'Plot' is also active. This output is necessary to create a loss field with spatial distribution. Such a loss field can be used alternatively to the above.

  Hint: Instead of 'Eddy Current Losses', we can also use 'Total Losses'. Such total losses contain eddy current losses plus core losses. To activate core losses, the material property Kh (Ke, Kx) must be set larger than zero. Kh models hysteresis losses resulting from changing magnetic fields.



4. Check the solution 'ThermalFlowSteadyState'.

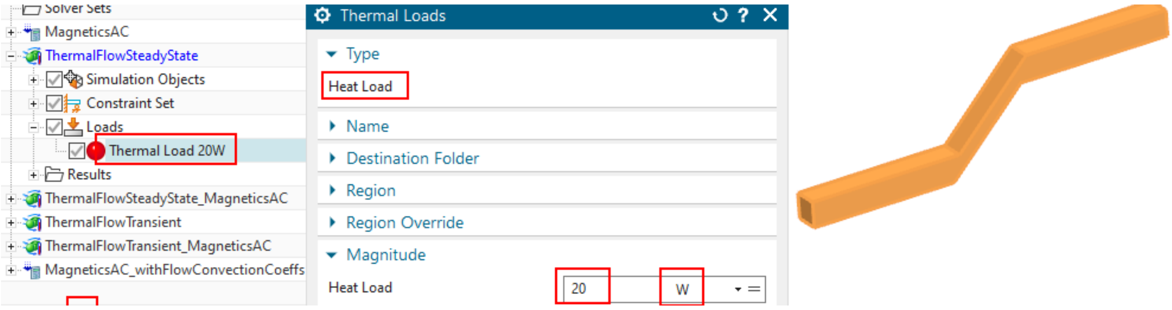
   - This is a coupled Thermal/Flow solution. So, it computes temperatures, fluid velocities and pressures.
   - There is a inlet flow velocity of 5 m/s defined in z direction what leads to a cooling effect.
   - In the 'Results Options', '3D Flow', there are the convection coefficients requested.
   - the Fem file contains both Magnetics and Thermal/Flow properties. The Magnetics properties can be seen, when the solver is set to MAGNETICS (Fem part, Edit) and the Thermal/Flow properties can be seen, when it is set to that solver.
   - We will solve this steady state solution but notice, there is also a transient solution 'ThermalFlowTransient' that can also be solved to simulate transient heating.

5. Solve the solution 'MagneticsAC'.

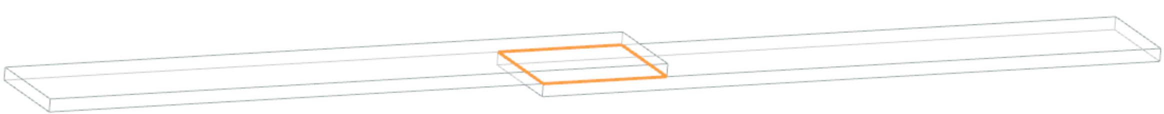
   - After the solve has finished, check the requested loss on the conductor in the corresponding text file '*.EddyCurrentLosses.txt'. It is approximately 20 W.
   - If there are any 2D conductors (electric interface resistances or conducting sheets) in the model, the text file would also contain the losses values for these.

6. Transfer losses, use integral value (**Alternative 1**)

- Activate solution 'ThermalFlowSteadyState' and check the existing load 'Thermal Load 20 W'. This is a 'Thermal Load' of type 'Heat Load', expecting the unit W and it is assigned to the solids. Therefore, this load type is convenient to transfer the integral loss value on solids from Magnetics.

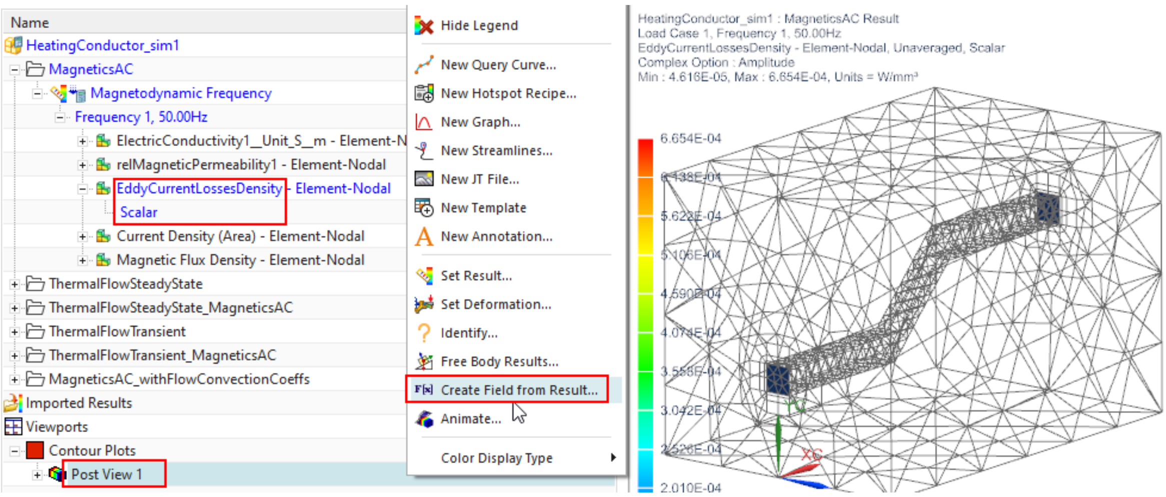


- If there are any 2D conductors (electric interface resistances or conducting sheets) in the model (not in the tutorial model, see picture below for an example), these would need additional loads assigned to those faces with their corresponding loss values.

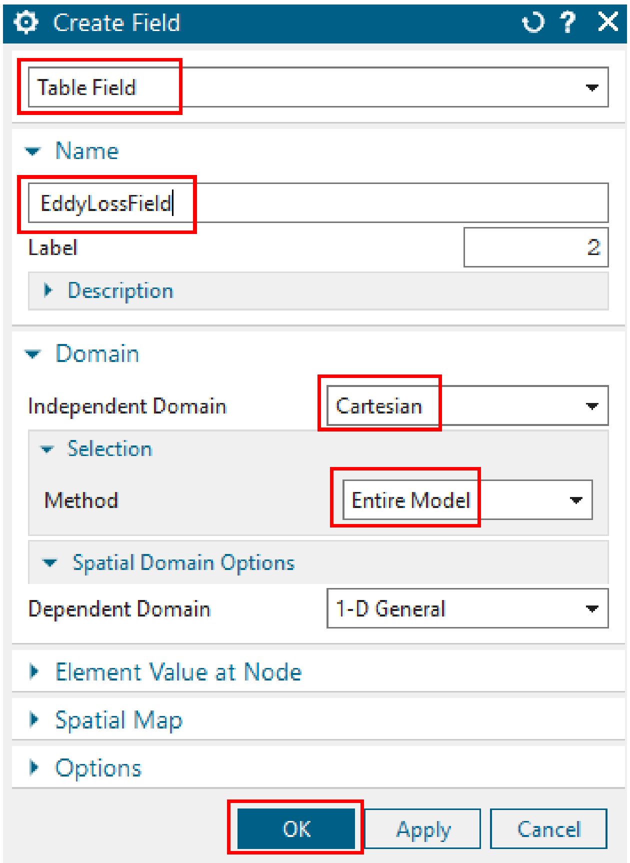


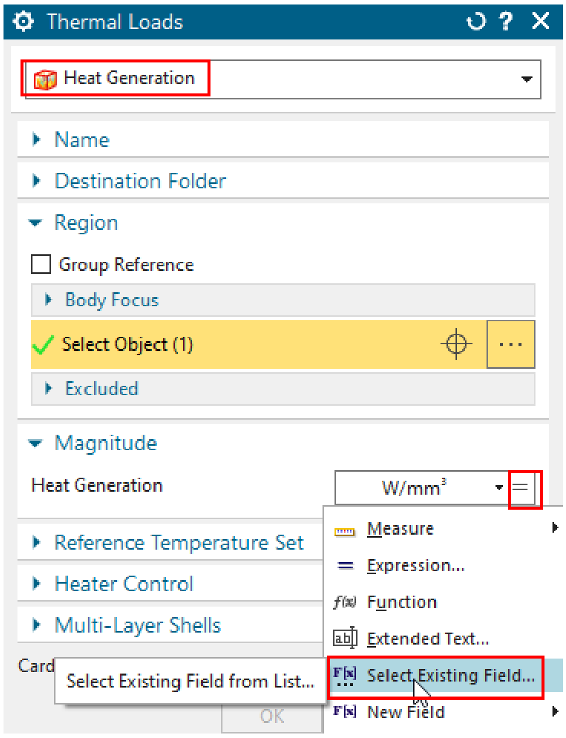
7. Transfer losses, use spatial field (**Alternative 2**).
While the above method (integral loss value) does not capture any spatial distribution of losses, there does exist an alternative way that overcomes this issue: The losses density can be stored as a spatial field in Simcenter. Then, in Thermal/Flow environment, such loss field is referenced in the heating load. Proceed as follows for this alternative:

- Display the magnetic losses result on solids (either 'EddyCurrentLossesDensity' or 'TotalLossesDensity') in a post view (see picture below).
- If there are any 2D electric interface resistances or conducting sheets in the model:
  - There will be an extra result 'EddyCurrentLossesDensityArea', unit W/mm^2.
  - That result must be stored in a separate field because of the different unit.
- Right mouse button on the 'Post View', choose 'Create Field from Result'.



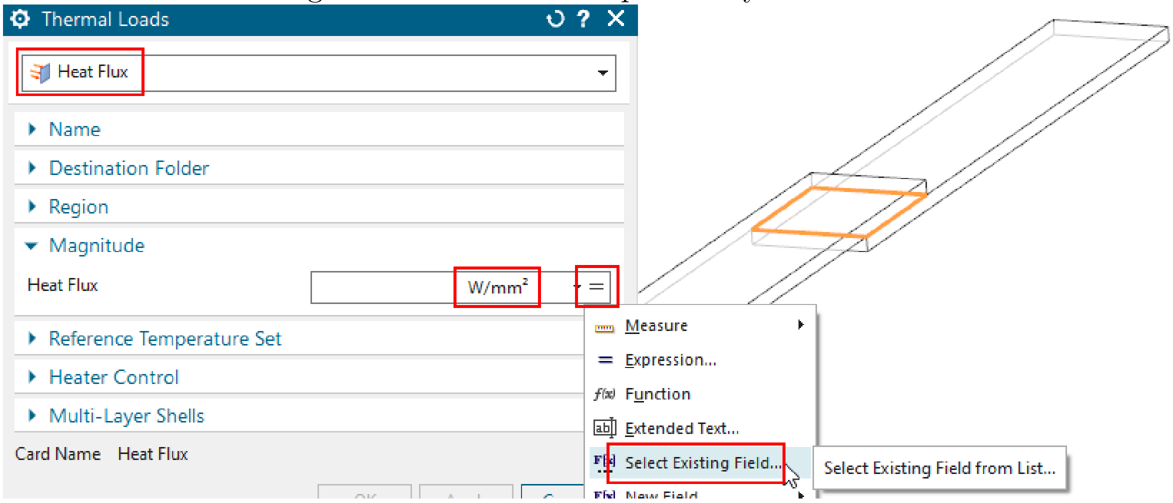
- The dialogue 'Create Field' appears. See picture below left side.
- Key in for name 'EddyLossField' (or similar). All defaults can be accepted.
  - Hint: Accept the default 'Independent Domain' of 'Cartesian'. This setting allows using different meshes for the electromagnetic and the thermal solutions, because interpolation is used.
  - Hint: Accept the default 'Selection Method' 'Entire Model'. Because the air does not have any loss results, it will not be written into the field.
- Click OK and the field is created.



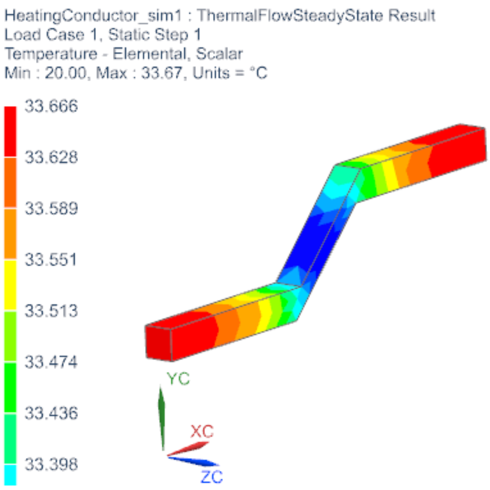


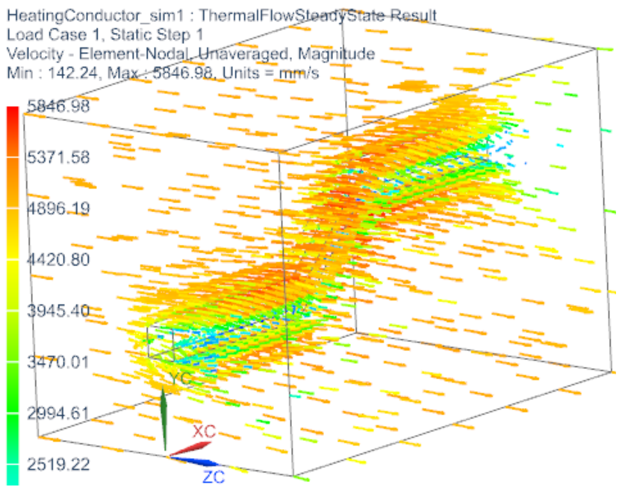
- Activate solution 'ThermalFlowSteadyState'. (Remove the existing load 'Thermal Load 20W if working on the tutorial model)
- Create a new 'Thermal Load'. See picture above right side. Set the type to 'Heat Generation'. This type expects the unit W/mm^3 that corresponds to the Magnetics losses density on solids.
- Select all solid conductor bodies.
- Instead of keying in a single loss value, choose 'Select Existing Field' and in the following window select the previously created field (EddyLossField). Click OK, OK.
- In case there are any 2D interface resistances or conducting sheets in the model:
  - This is not in the tutorial model, see picture below for an example.
  - There should already be a separate field previously created for their losses.
  - Create an additional 'Thermal Load' but now use the type 'Heat Flux' because this expects the unit W/mm^2 that corresponds to the 2D losses.

– Select the faces of such 2D interface resistances or conducting sheets.
– Choose 'Select Existing Field' and select the previously created field.



8. Solve the solution 'ThermalFlowSteadyState'.
   - After the solve has finished, feel free doing any post processing.
   - Following picture left shows the computed solid temperatures with a maximum of 33.6^0C and right the fluid velocity.





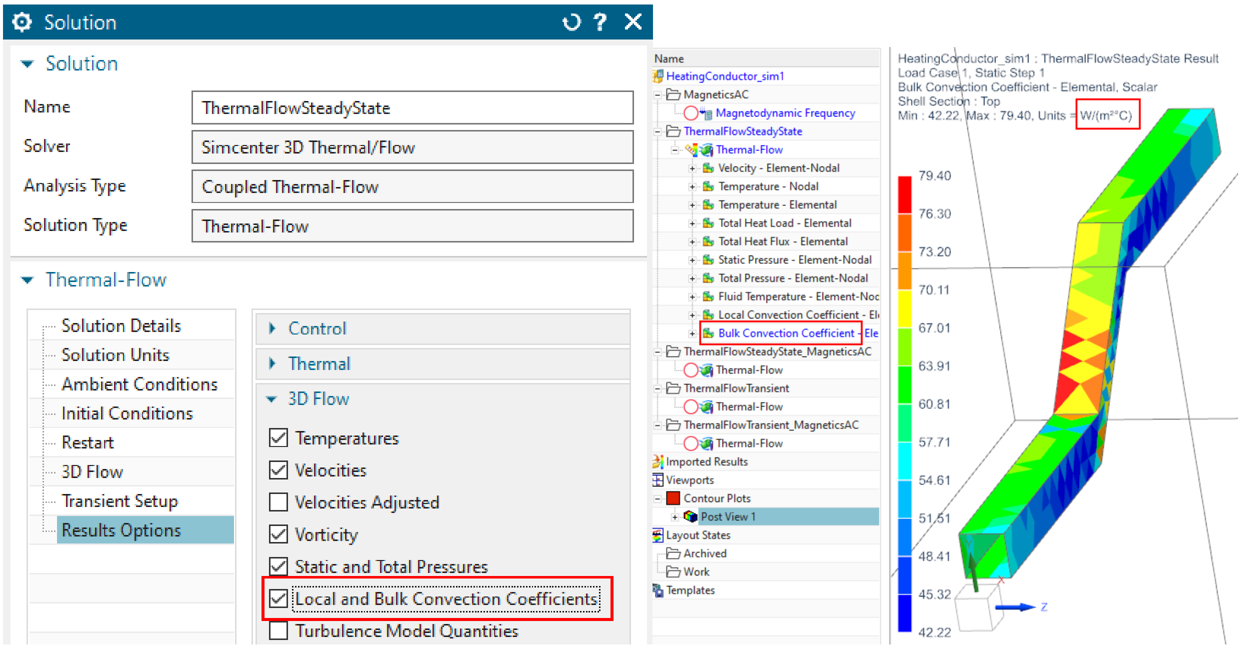
   - The advantage of this type of solution becomes clear: The flow, coming from the side, leads to a conductor cooling that can not be easily modeled by fixed convection effects. Thus, a pure Magnetics solve cannot capture this. One possibility to overcome this is shown in the following chapter: 'One point five Way Coupling via Convection Coefficients'.

9. The tutorial is finished.

2.1.2 One point five Way Coupling via Convection Coefficients

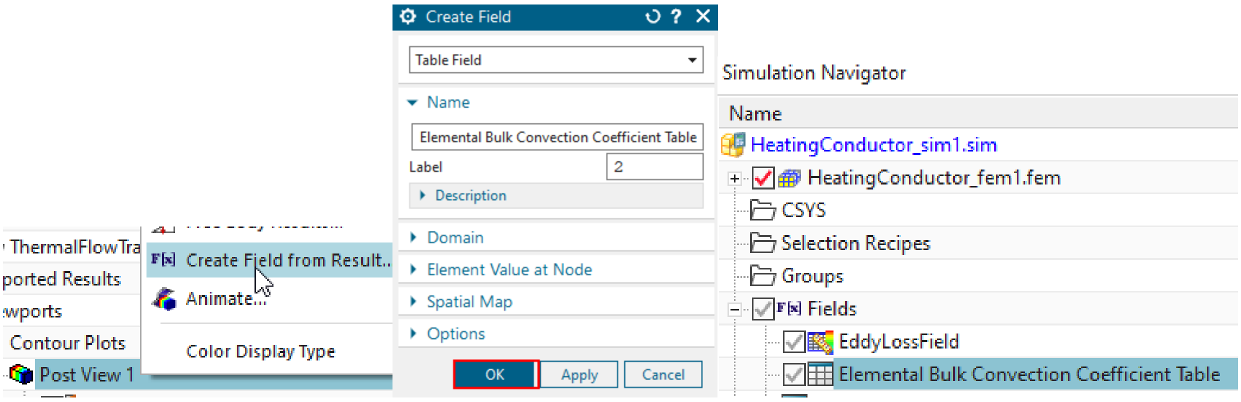
The Simcenter Thermal/Flow(ESC) solver has the powerful capability of finding accurate local heat transfer convection coefficients (HTC). At each element face of a fluid-solid interface, the CFD method calculates such an coefficient, taking into account the local fluid velocity, turbulence characteristic and fluid temperature. Such coefficients can be computed once by the Thermal/Flow(ESC) solver and then reused by NX Magnetics' internal thermal solver. This allows doing different kinds of EM/Thermal simulations all in NX Magnetics, without the need for the Thermal/Flow solver any more. Following, we show how this is done.

- We begin from the model of the previous chapter. Open the Sim file 'HeatingConductor_sim1.sim'. This model contains Magnetics and Thermal/Flow solutions from the previous chapter.
- Activate the solution 'ThermalFlowSteadyState'.
- Activate the output of the thermal convection coefficients: Edit the solution and in register 'Results Options' in Box '3D Flow', activate 'Local and Bulk Convection Coefficients' (see picture below, left). In the tutorial model, the button is already activated.
- Hint: The bulk convection coefficients are related to the ambient temperature (usually 20^0C). And the local coefficients are related to the local fluid temperature near the wall. So, we will use the bulk type because we know the ambient temperature only.



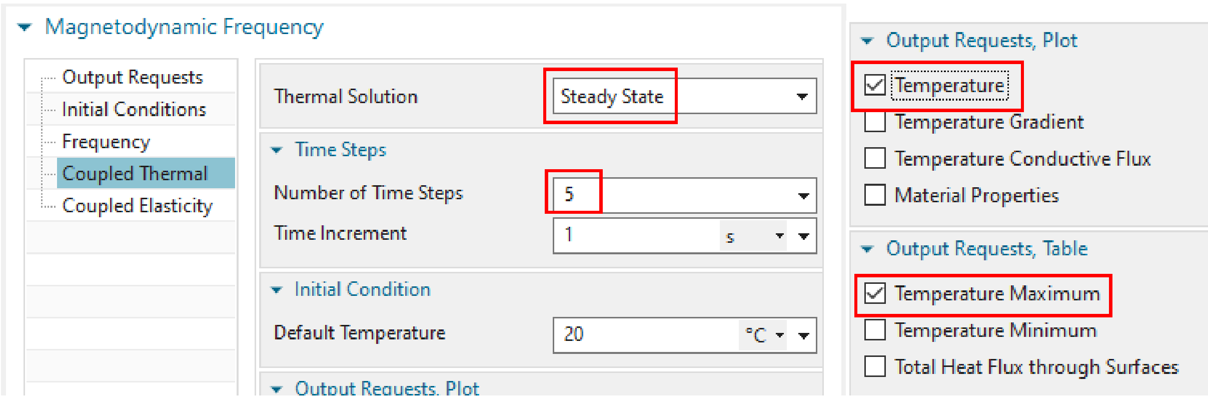
- Solve the model and display the result 'Bulk Convection Coefficient'.
- The convection coefficients, as seen in picture above, right are shown in SI units $W/(m^2C)$. In 'Edit Post View', 'Result' the unit can be changed. In this example, they vary between 42 and 79.
- Create a field from this result: Edit the Post View and select 'Create Field from Result'. In the following dialogue 'Create Field', accept all defaults and click 'OK'. The Simulation

Navigator shows the newly created field (picture below).



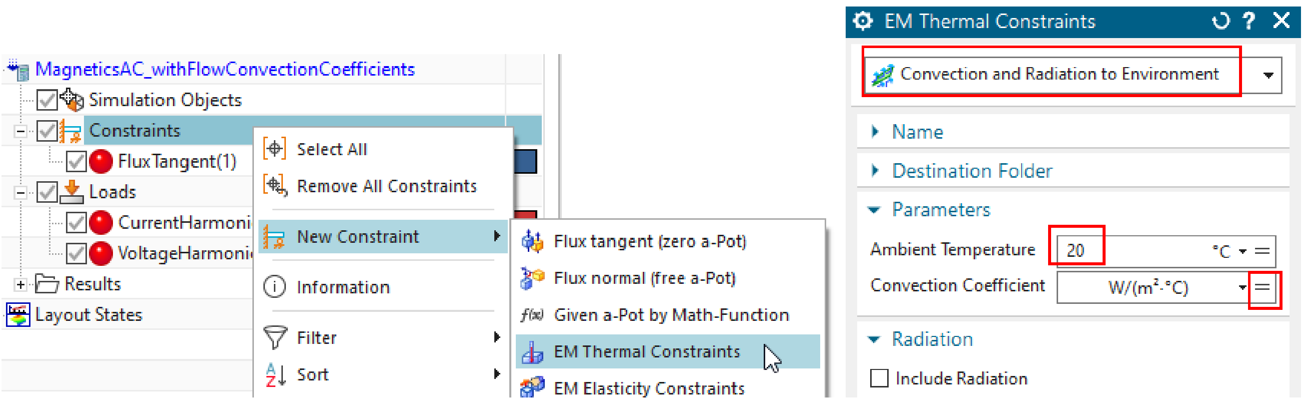
Now, the field with accurate convection coefficients can be used in NX Magnetics for coupled thermal solutions. To do so, proceed as follows:

- Edit the solution 'MagneticsAC_withFlowConvectionCoeffs' and in register 'Coupled Thermal', set the 'Thermal Solution' to 'Steady State' (or transient if desired). Set the 'Number of Time Steps' to 5 to allow material property updates. Activate the 'Temperature' plot output request and the 'Temperature Maximum' table output request. These parameters are already set.



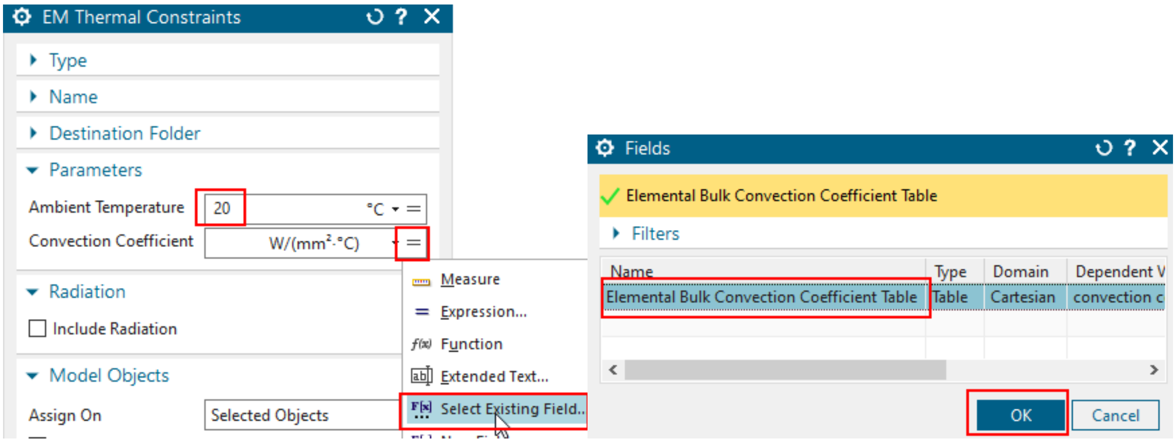
- Delete the constraint 'Thermal Convection with Coefficients Field'. We will create this one in the next steps.

- Create a constraint of type 'EM Thermal Constraints'. Set the type to 'Convection and Radiation to Environment'.

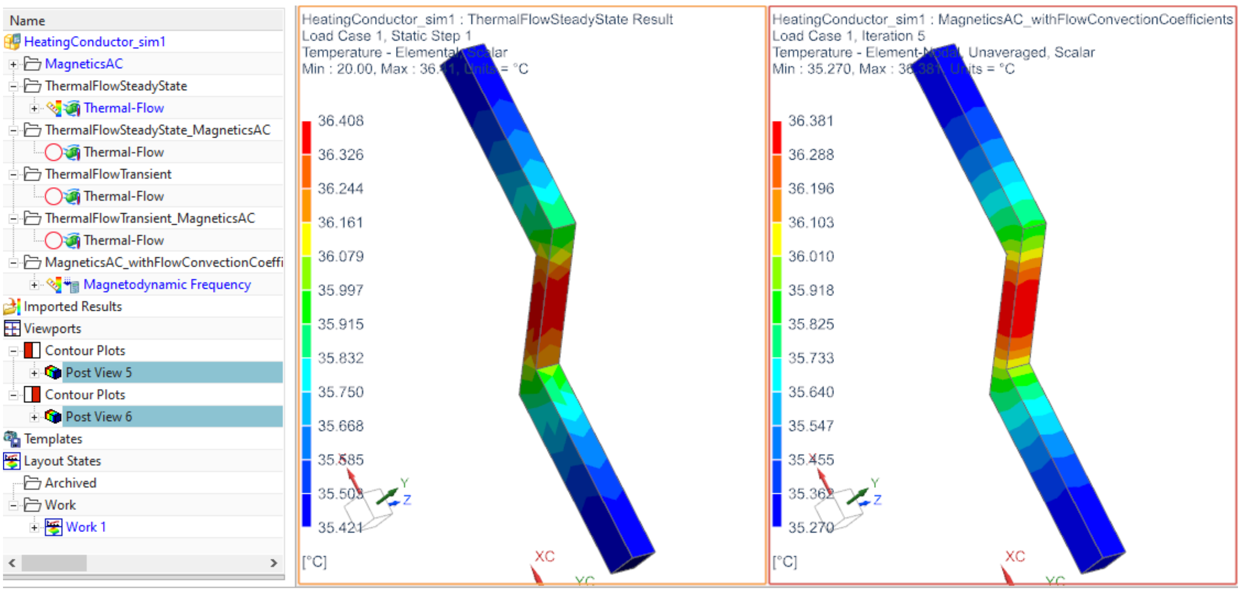


- Blank all meshes and the Air body and drag a window over all faces of the conductor. Deselect the two electrode faces.
- Key in the same 'Ambient Temperature', that has been used in the flow solution (20^0C), that was used to find the convection coefficients. This step is important because the convection coefficients are only valid for one ambient temperature.
- At 'Convection Coefficient', click on the = symbol and select 'Select Existing Field'. In the following window, select the newly created field with the convection coefficients, Click OK, OK to finish the process.

  Hint: The units are automatically set to those of the field. No need to change them here.



- Solve the solution and display the temperature result. Compare the result with that of the Thermal/Flow solution 'ThermalFlowSteadyState' (that gave us the convection coefficients). Both solutions should show results in good agreement. The picture below shows left the Thermal/Flow and right the NX Magnetics result. The agreement of the two results becomes even better, if the Thermal/Flow used the eddy current losses spatial field (above called as Alternative 2).



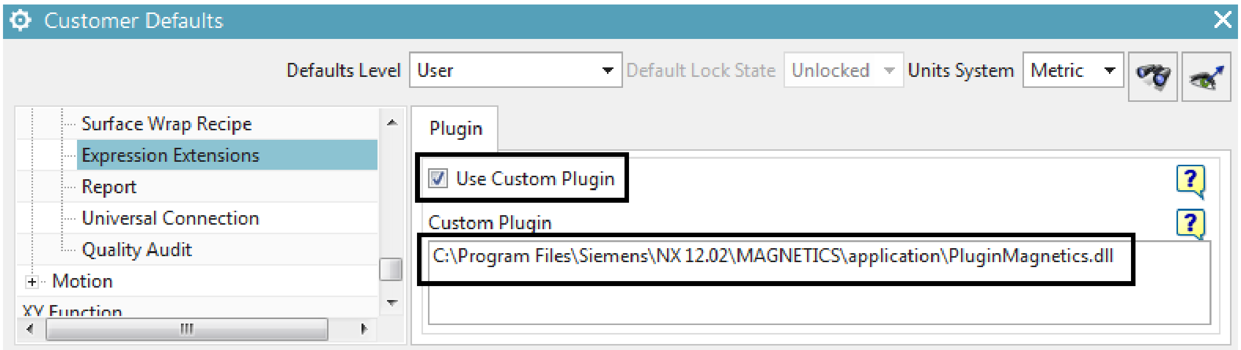
Advantages of this kind of 1.5 way coupling arises as soon as following simulations are required. For example, we want to do transient heating of the conductor. As long as the outside flow stays the same, convection characteristic (and thus the coefficients) will be influenced only very little from changes in the electromagnetic solution. Therefore, we can reuse the field with the convection coefficients and don't need updates to the costly Thermal/Flow solution.

The tutorial is complete.

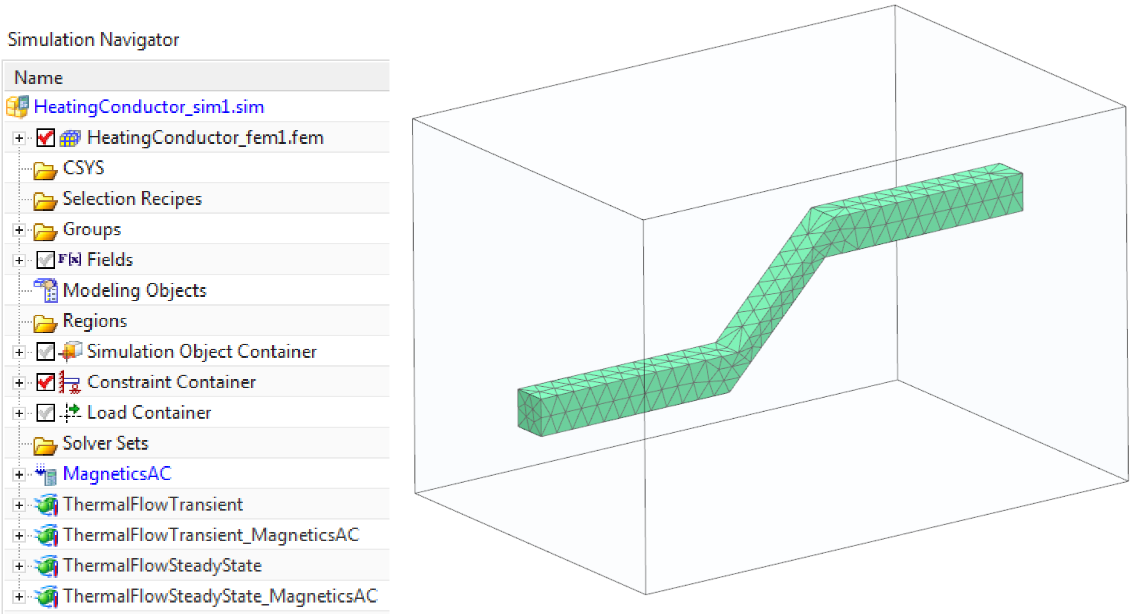
2.1.3 Two Way Coupling via Plugin Solver

This example demonstrates heating of conductors with use of Simcenter Thermal/Flow (or ESC) solvers and Magnetics in deep integration. Therefore Thermal/Flow computes temperature fields and flow velocity using eddy current losses as input. Magnetics updates material properties with new temperatures and computes for new electromagnetic fields and eddy current losses. The two solvers run alternating, controlled by Thermal/Flow. So Thermal/Flow acts as the master and calls Magnetics after a defined number of iterations.
The model is already build and ready to solve. In this tutorial we go through the existing features to check and explain them. Follow the steps.

1. Start NX / Simcenter in version 12 or later.
2. Activate the Plugin (necessary only once)
   - Open the Customer Defaults (File→Utilities→Customer Defaults)
   - Navigate to Simulation→Pre/Post→Expression Extension.
   - Activate 'Use Custom Plugin' and key in the full path and file name corresponding to your Magnetics installation as shown in the picture below.

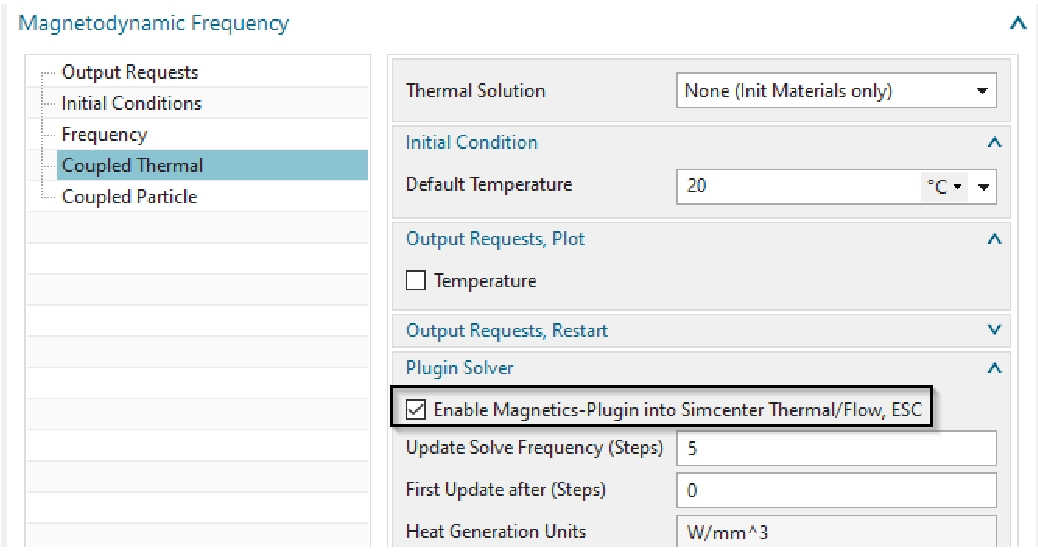
   

   - Click OK and restart Simcenter to make the modification active.
3. Download the model files for this tutorial from the following link:
   `https://www.magnetics.de/downloads/Tutorials/7.CouplThermal/7.2HeatingCond_CoupledFlow.zip`
4. Open the Sim file 'HeatingConductor_sim1.sim'.



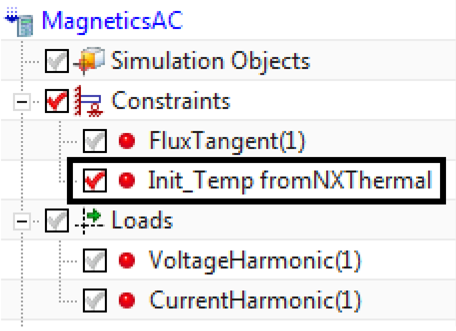
5. Check solution 'MagneticsAC'

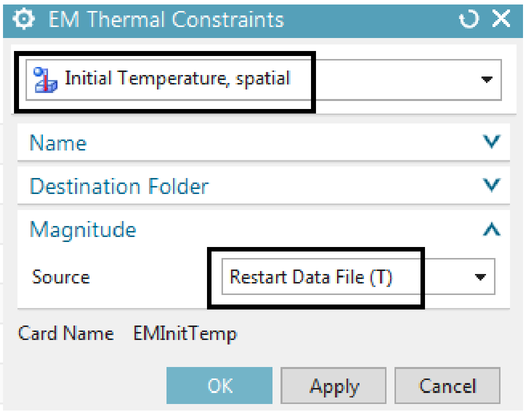
   - Edit the solution. Click on register 'Coupled Thermal'
   - Notice in box 'Plugin Solver' the 'Enable Magnetics-Plugin ...' button is activated. Through this setting Magnetics will write eddy current losses into a file (.P.pos) that will be used by Thermal/Flow. Also Magnetics will write a file (.Plugin.ini) that is used to control the iterative process.
   - Notice in the same box the setting 'Update Solve' is set to 5. This means Magnetics will be triggered every fifth time step from Thermal/Flow.



   - Close the window.
   - Check the constraint named as 'Init_Temp fromNXThermal'. This is a 'EM Thermal Constraint' of type 'Initial Temperature, spatial'. It allows defining initial temperatures as a spatial field. Through this feature the Thermal/Flow solver will transfer the resulting temperature distribution into Magnetics. Magnetics will use these temperatures to update the materials with those temperatures and perform a new solve.

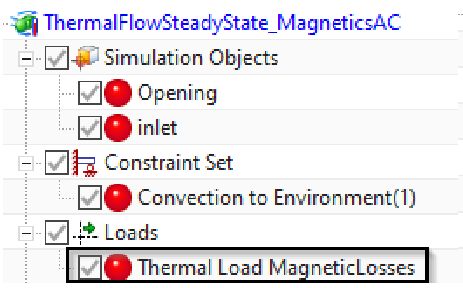
This feature works similar as a reference-field, but it is more direct between the two solvers.

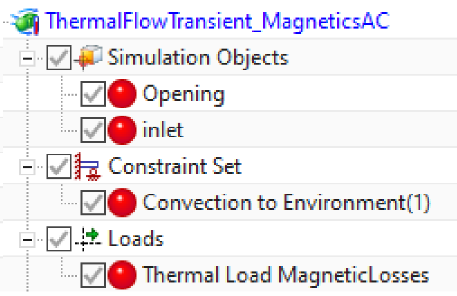




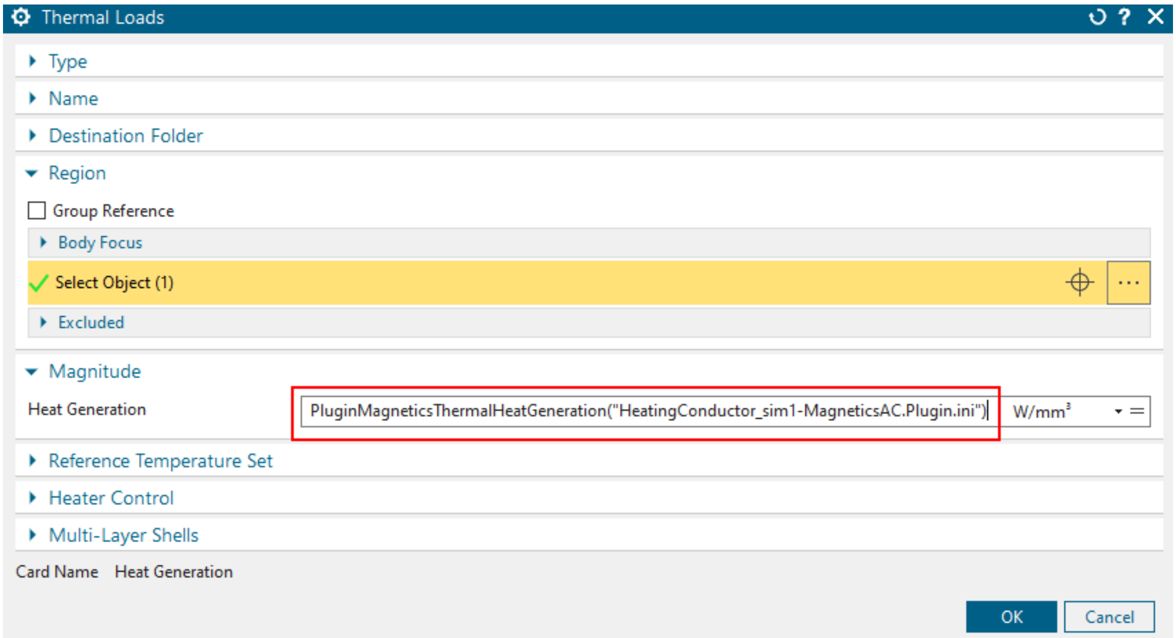
6. Check solution 'ThermalFlowSteadyState_MagneticsAC'

- Activate the solution and check settings as desired. There is nothing special in this solution, all settings are defaults.
- While this is a steady-state solution, there is also a transient solution named 'ThermalFlowTransient_MagneticsAC' available (picture below right) that can be used to solve for the transient heating with Magnetics update.





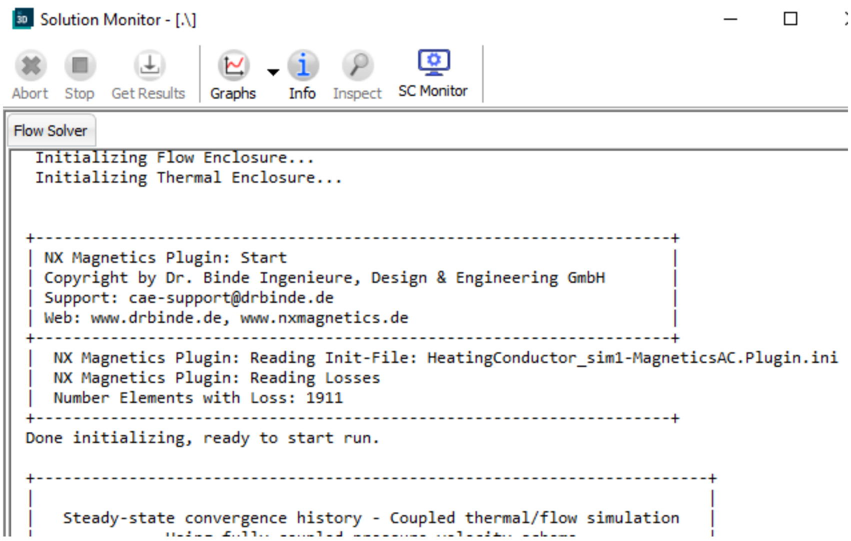
- Check the load named 'Thermal Load MagneticLosses' (see picture below). This is a load of type 'Heat Generation'. It expects the load in unit W/mm^3.
- Notice, instead of a fixed load value, there is a plugin function assigned to that load. That plugin function is named 'PluginMagneticsThermalHeatGeneration' and it takes as argument the name of a control file pointing to the Magnetics solution: 'HeatingConductor_sim1-MagneticsAC.Plugin.ini'. Hint: This ini file is automatically created from the Magnetics solve.
- In this load, there must be all conductor bodies selected (only one in our case). Thus, the eddy current losses are applied here only.
- The two solvers communicate by this feature.



- Close the window.

7. Solve solution 'MagneticsAC'.
8. Solve solution 'ThermalFlowSteadyState_MagneticsAC'.

- The solution monitor shows the start of the Magnetics plugin.



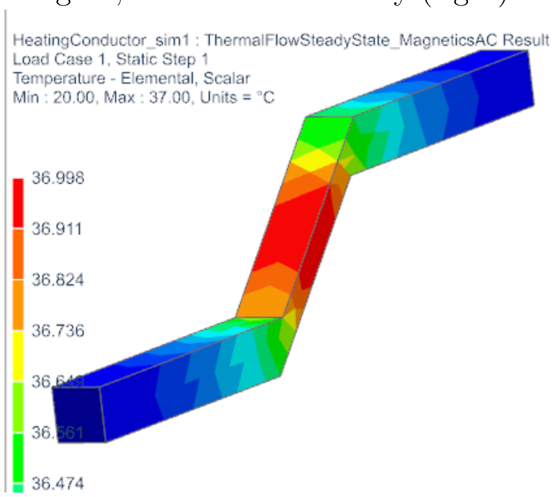
- Then, every 5th iteration, Magnetics is called to update the solution.

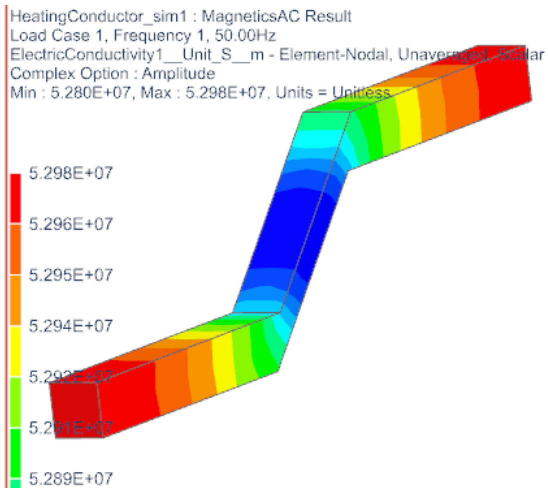
```
 | H - Energy       |    2  |   1.610e+03  |  0.56  |   2.576e-02  |  --  |
 +------------------+-------+--------------+--------+--------------+------+
 | Thermal Enclosure - Step 4                                              |
 +------------+-----------+------------+-----------+------------+----------+
 | CSOLVER    |  Minimum  | Location   |  Maximum  | Location   | Average  Convg|
 +------------+-----------+------------+-----------+------------+----------+
 | y+         | 1.09E+00  |       9522| 1.41E+01  |       9482| 8.47E+00   LO |
 | T-Solid    | 2.00E+01  |       9054| 3.70E+01  |       1224| 3.35E+01   OK |
 +------------+-----------+------------+-----------+------------+----------+

 +------------------------------------------+
 |           Coupled thermal convergence     |       Heat balance
 +------------------+-------+----------+------+ -------------------------
 |    Criteria      | Rate  |  |Change|  | Msg  | To fluid:   2.308e+07
 +------------------+-------+----------+------+            (2.298e+07)
 | Heat Imbalance   | 8.57  |  4.351e-03 |  OK  | To solid:   0.000e+00
 | Max delta T      | 0.44  |  5.883e-01 |  --  |            (0.000e+00)
 +------------------+-------+----------+------+
 | NX Magnetics Plugin: Solver Start
Info    : Started (Tue Nov 30 10:14:28 2021, Wall = 0.00699997s, CPU = 0.015625s, Mem = 8.02734Mb)
P r e - P r o c e s s i n g . . .
Info    : System 1/2: 544 Dofs
Info    : System 2/2: 35626 Dofs
P r e - R e s o l u t i o n  (1/1) . . .
Info    : System 1/1: 544 Dofs
E n d   P r e - R e s o l u t i o n  (1/1)
E n d   P r e - P r o c e s s i n g
P r o c e s s i n g . . .
Solving Frequency 50 Hz
------------------ Successfully Ended ------------------
E n d   P r o c e s s i n g
Info    : Stopped (Tue Nov 30 10:14:31 2021, Wall = 2.602s, CPU = 2.98438s, Mem = 292.801Mb)
 | NX Magnetics Plugin: Solver Command returned: 0 (0: Ok)
Magnetics calculation time: 2896
 | NX Magnetics Plugin: Reading Losses
 | Number Elements with Loss: 1911
 +------------------------------------------------------------------+

                    +------------------------+--------------------------------+
 Global iteration  |     Linear Solver      |       Convergence info         |
        5          +------------------------+--------------------------------+
                    | Iters |    |Ax-b|     | Rate  |   Residual    |  Msg  |
```

- After the solve has finished, feel free doing any post processing.
- Following picture (left) shows the solid temperatures with a maximum of 37 ^{0}C.
- The picture right shows the resulting electric conductivity in unit S/m. This is computed from the magnetic solution and it is displayed because the output request 'Material Properties' is activated. Verify from the two pictures: If temperature (left) is higher, electric conductivity (right) is smaller.





The tutorial is finished.

2.1.4 Two Way Coupling via Manual Process

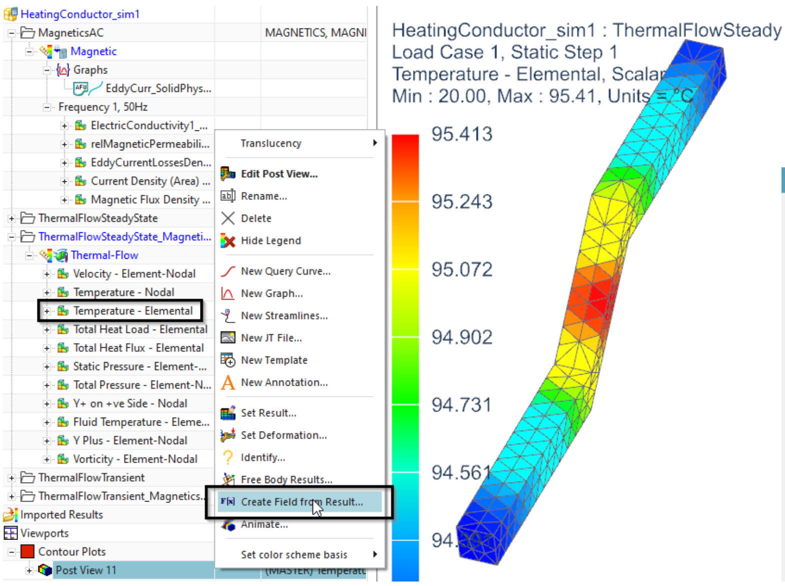
This chapter demonstrates another alternative for the heating of conductors with use of Simcenter Thermal/Flow (or ESC) solvers and Magnetics. The coupling is carried out in a manual way while the previous chapter showed a plugin feature to perform this automatized. The manual process runs as follows:

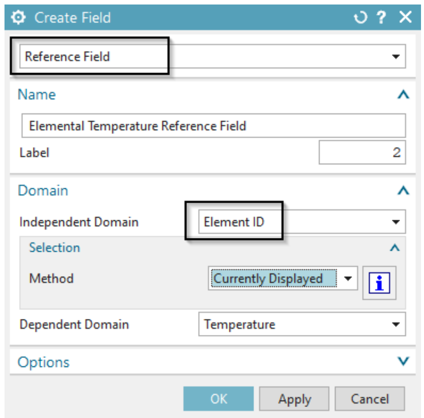
1. The Magnetics solve is first performed. The thermal initial condition is default, e.g. all materials have 20^0C. The losses result is stored in a reference field (This approach is demonstrated in the previous chapter: 7. Transfer Losses, use Field).

2. Next the Thermal/Flow or ESC solver runs. It uses the previously defined reference field with losses as load (as shown before). The resulting temperatures on the conductor are stored in another reference field. This second field must be a 'Elemental Temperature Reference Field' and it is created as follows.

   (a) In the post processor display the result type 'Temperature Elemental'. See picture below left.

   (b) RMB on the 'Post View' choose 'Create Field from Result'. The dialogue 'Create Field' appears. Set the option to 'Reference Field' and click 'OK'. See picture below right.





3. Now, to perform an update, the Magnetics solver gets a initial thermal constraint that uses the (second) reference field containing the temperature field. Using this initial condition the Magnetics solver is started and after finishing the updated losses are automatically stored in the existing (first) reference field.

4. From now on the user can alternately start the two solvers. The maximum temperature should increase slightly and with each iteration this increase will reduce, ending in a converged situation.

Therefore Thermal/Flow computes temperatures in the conductor and flow velocity using eddy current losses as heating load. Magnetics reads the temperature field on the conductor and updates material properties for those temperatures. Following Magnetics computes new electromagnetic fields and eddy current losses. The two solvers run alternating, controlled by the user.

The tutorial is finished.

3 Features

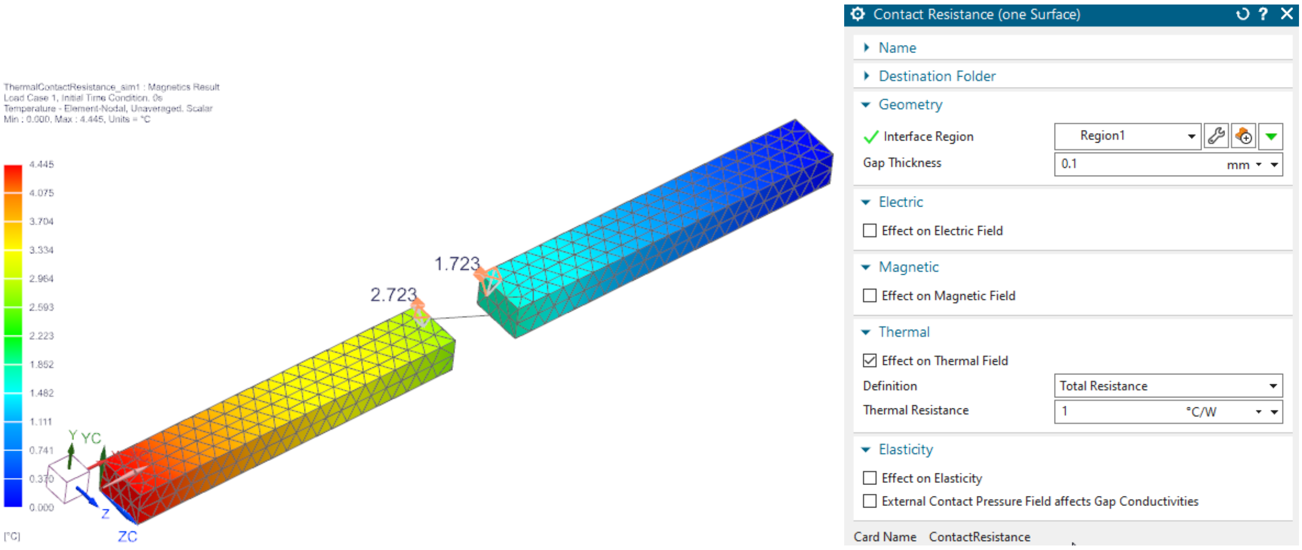
Additional features to be used in several solutions are demonstrated in this chapter.

3.1 Contact Resistance - Thermal

This is a Thermal Testcase with FluxCoupling and ContactResistance. We apply 1 W and a resistance of $1W/C$. Therefore, we expect a temperature difference of 1^2C at the interface. The simulation result agrees well (picture below).

Download the model files for this tutorial from the following link:
`https://www.magnetics.de/downloads/Tutorials/10.Features/10.4ThermalInterfaceResistance.zip`

The following figure shows the two conductors. On the left and right electrode faces there are fixed temperatures (zero and 100 degrees) applied.



The picture also shows the dialogue of the simulation object that is used to create such an 'Contact Resistance'