

# Electromagnetic Basic Tutorials for NX-Magnetics

Dr. Binde Ingenieure

June 30, 2024

2011-2025 Dr. Binde Ingenieure, Design & Engineering GmbH. All Rights Reserved. This software and related documentation are proprietary to Dr. Binde Ingenieure, Design & Engineering GmbH. All other trademarks are the property of their respective owners.

DR. BINDE INGENIEURE, DESIGN & ENGINEERING GMBH MAKES NO WARRANTY WHATSOEVER, EXPRESSED OR IMPLIED THAT THE PROGRAM AND ITS DOCUMENTATION ARE FREE FROM ERRORS AND DEFECTS. IN NO EVENT SHALL DR. BINDE INGENIEURE, DESIGN & ENGINEERING GMBH BECOME LIABLE TO THE USER OR ANY PARTY FOR ANY LOSS, INCLUDING BUT NOT LIMITED TO, LOSS OF TIME, MONEY OR GOODWILL, WHICH MAY ARISE FROM THE USE OF THE PROGRAM AND ITS DOCUMENTATION.

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

<b>1</b>	<b>Introduction</b>	<b>5</b>
1.1	Working with the Tutorials . . . . .	5
1.2	Electromagnetic Principles . . . . .	6
1.2.1	Maxwell Equations . . . . .	6
1.2.2	Material Equations . . . . .	6
1.2.3	Electromagnetic Models . . . . .	7
<b>2</b>	<b>Recommended System Settings</b>	<b>9</b>
2.1	Set the Role to Advanced . . . . .	9
2.2	Set Material Library . . . . .	9
2.3	Create Non-Manifold Polygon Bodies . . . . .	10
2.4	Use Polygon Body Names for Meshes and Physicals . . . . .	12
2.5	Use High Resolution for Polygon Bodies . . . . .	13
<b>3</b>	<b>CAD Basics</b>	<b>14</b>
3.1	3D Solenoid (Beginner) . . . . .	14
3.1.1	Preparations . . . . .	14
3.1.2	Sketch the Core . . . . .	15
3.1.3	Extrude the Core . . . . .	16
3.1.4	Zoom, Rotate and Pan . . . . .	17
3.1.5	Sketch the Iron . . . . .	17
3.1.6	Extrude the Iron . . . . .	18
3.1.7	Sketch the Coil . . . . .	19
3.1.8	Extrude the Coil . . . . .	21
3.1.9	Create a Spherical Air Volume . . . . .	22
3.1.10	Assign Names to the Bodies . . . . .	23
<b>4</b>	<b>Magnetostatic</b>	<b>25</b>
4.1	Core and Inductor (Beginner) . . . . .	26
4.1.1	Preparations . . . . .	26
4.1.2	Fem and Sim File Creation . . . . .	27
4.1.3	Fem File Steps . . . . .	29
4.1.4	Sim File Steps . . . . .	35
4.1.5	Axisymmetric Analysis of Core and Conductor . . . . .	39
4.2	Static Force on Solenoid (TEAM 20) (Beginner) . . . . .	41
4.2.1	Preparations . . . . .	41
4.2.2	Create Fem and Sim Files . . . . .	43
4.2.3	Create the Solution . . . . .	44
4.2.4	Parametric Meshing . . . . .	44
4.2.5	Define Core/Iron Material with Nonlinear B-H Curve . . . . .	46
4.2.6	Define Stranded Coil Properties . . . . .	47
4.2.7	Define Infinity Properties . . . . .	49
4.2.8	Finish Fem File . . . . .	50
4.2.9	Define Load and Constraint . . . . .	51
4.2.10	Solve and Post Processing . . . . .	52

4.2.11	Increasing Accuracy by Reducing Nonlinear Residuals . . . . .	54
4.2.12	Increasing Accuracy by global higher Order Shape Functions . . . . .	54
4.2.13	Increasing Accuracy by local higher Order Shape Functions . . . . .	55
4.2.14	Increasing Accuracy by Adaptive Meshing . . . . .	57
4.3	Force on Magnet (TEAM 23) . . . . .	59
<b>5</b>	<b>Magnetodynamic</b>	<b>61</b>
5.1	Nonlinear Transient Rotational Test Rig (TEAM 24) . . . . .	62
5.1.1	Strategy . . . . .	62
5.1.2	Set up the Fem Model . . . . .	62
5.1.3	Find Coil Fillfactor, Rerun and Test . . . . .	70
5.1.4	Post Processing . . . . .	71
5.2	Inductance, Phase Shift and Resistance of a Coil in 1D,2D,3D . . . . .	74
5.2.1	1D Circuit Simulation . . . . .	74
5.2.2	Post Processing . . . . .	76
5.2.3	Analytic Verification . . . . .	76
5.2.4	2D Axisymmetric Analysis . . . . .	77
5.2.5	Post Processing . . . . .	80
5.2.6	3D Analysis . . . . .	81
5.2.7	Post Processing . . . . .	84
5.2.8	Result Comparison with Analytic . . . . .	85
5.3	Conductive Plate under Coil (TEAM 3) . . . . .	87
5.3.1	Analysis of Complete Geometry . . . . .	87
5.3.2	Post Processing Complex Results . . . . .	92
5.3.3	Modeling the Plate by 2D Elements . . . . .	95
5.3.4	Using Symmetry . . . . .	98
5.4	Shielding in Time Dependent Magnetic Field (TEAM 1a) . . . . .	99
5.4.1	Solution with Potential Condition . . . . .	100
5.4.2	Solution with Electric Current Load . . . . .	109
5.5	Resonant Circuit with 3D Coupling . . . . .	112
5.5.1	1D Circuit Simulation . . . . .	112
5.5.2	Post Processing . . . . .	113
5.5.3	Analytic Verification . . . . .	113
5.5.4	Circuit with 3D Resistance . . . . .	114
5.6	1D, 2D, 3D Connector . . . . .	116
5.7	Isolating Layer . . . . .	117
<b>6</b>	<b>Electrostatic</b>	<b>118</b>
6.1	Microstrip Line (Beginner) . . . . .	118
6.1.1	Preparations . . . . .	118
6.1.2	Fem File Steps . . . . .	119
6.1.3	Sim File Steps . . . . .	123
<b>7</b>	<b>Electrokinetic</b>	<b>125</b>
7.1	Circular Plate . . . . .	125
7.1.1	Preparation . . . . .	125
7.1.2	Sim File Check . . . . .	125

7.1.3	Post Processing . . . . .	126
7.1.4	Result Comparison with Analytic . . . . .	127
<b>8</b>	<b>Electrodynamic</b>	<b>128</b>
8.1	Capacitor . . . . .	128
8.1.1	Result Comparison with Analytic . . . . .	128
<b>9</b>	<b>More Features</b>	<b>129</b>
9.1	Contact Resistance - Electrical and Thermal . . . . .	130
9.2	Flux Coupling - Electrical and Thermal . . . . .	132
9.3	Surface to Surface Glue - Electromagnetic, Elastic and Thermal . . . . .	134
9.4	Orthotropic Materials Modeling . . . . .	135
9.5	Magnetic Hysteresis, Jiles-Atherton Model . . . . .	137
9.6	Post-only - Run separate Post-Processing . . . . .	141
<b>10</b>	<b>Error Diagnostics</b>	<b>143</b>
10.1	MUMPS Equation Solver Messages . . . . .	143
<b>11</b>	<b>Parallel Processing</b>	<b>146</b>
11.1	SMP on one Computer . . . . .	146
11.2	DMP on one Computer . . . . .	146
11.3	DMP on Computer Cluster . . . . .	146
11.4	SMP and DMP combined on Computer Cluster . . . . .	148
<b>12</b>	<b>References</b>	<b>149</b>

# 1 Introduction

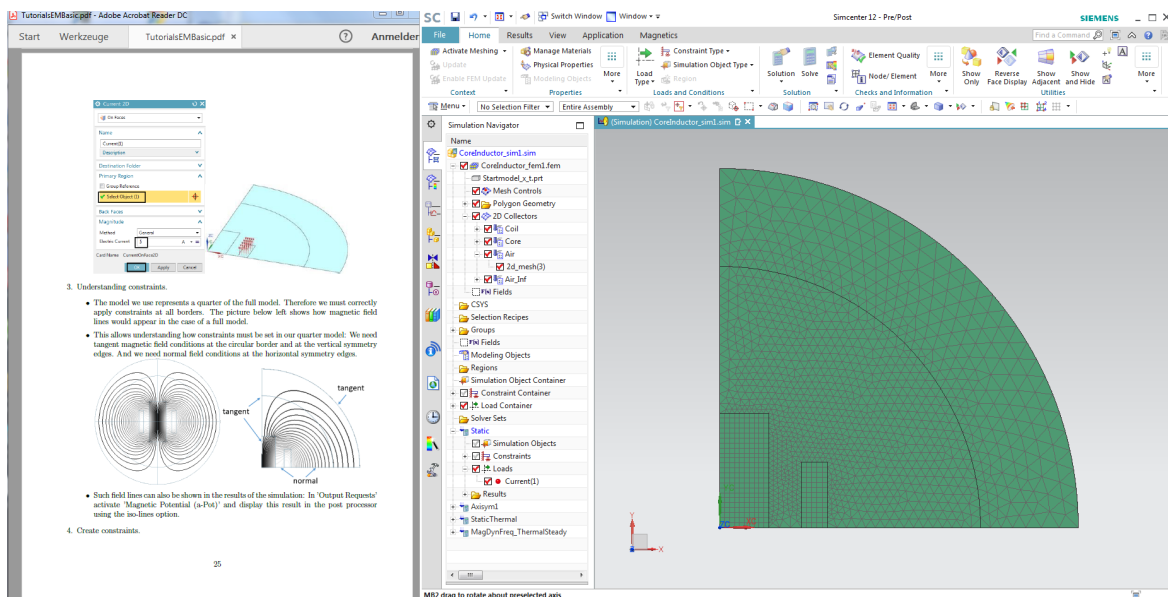
## 1.1 Working with the Tutorials

These tutorials shall allow the new user to quickly become familiar with MAGNETICS for Simcenter. Best would be to go through them in the order listed but it is also possible to start with any of them. The following three tutorials are especially convenient for beginners:

- 3D CAD Basics: 'Solenoid'
- 2D Simulation: 'Core and Inductor'
- 3D Simulation: 'Static Force on Solenoid'

The Simcenter/NX files for these tutorials can be downloaded as a zip-archive and the download-link is given at the beginning of each section. The files of each tutorial have a folder 'complete' that contains the files of the completed tutorial. Also a folder called 'start' exists, that contains the files to start with when going through the exercise.

We recommend, when working through the tutorials, positioning this pdf and the Simcenter window side by side on your screen (see picture below).



Now we wish all users a lot of success in learning and performing electromagnetic simulations with MAGNETICS for Simcenter 3D or NX.

In case of problems please write to the support address [cae-support@drbinde.de](mailto:cae-support@drbinde.de) and our team will be there for you.

With best wishes, your team of Dr. Binde Ingenieure

## 1.2 Electromagnetic Principles

This section is based on the lecture notes on 'Applied & Computational Electromagnetics' from the University of Liège [Geuzaine 2013], on [Binde] and on [Kost].

In the context of electromagnetic fields, Maxwell's equations are solved. Depending on the application, e.g. electrostatic or magnetodynamics, the equations are simplified or only subsets are considered. For the solution of the equations, the finite element method has been established.

In the following, we provide the usual governing electromagnetic equations, Maxwell's equations and material relations that are the foundations of said models. We then explain which type of model is the right one for a certain application problem, and finally provide the equations that belong to the individual models.

### 1.2.1 Maxwell Equations

Let's look at the Maxwell equations that describe the electromagnetic effects and are the basis for the models or applications listed above. Maxwell's equations are a set of four equations.

Equation name	Differential form	Remarks and units
Ampere's law	$\text{rot } \mathbf{H} = \nabla \times \mathbf{H} = \frac{d\mathbf{D}}{dt} + \mathbf{J}$	Electric current $\mathbf{J}[A]$ creates a rotating magnetic field $\mathbf{H}[A/m]$ . If electric fluxdensity $\mathbf{D}[As/m^2]$ changes with time, there will also be a rotating magnetic field created.
Faraday's law	$\text{rot } \mathbf{E} = \nabla \times \mathbf{E} = -\frac{d\mathbf{B}}{dt}$	If magnetic fluxdensity $\mathbf{B}$ changes with time, there will be a rotating electric field $\nabla \times \mathbf{E}$ created.
Gauss's law	$\text{div } \mathbf{D} = \nabla \cdot \mathbf{D} = \rho$	Electric charge $\rho$ is a source for electric fluxdensity $\mathbf{D}$ .
Gauss's magnetic law	$\text{div } \mathbf{B} = \nabla \cdot \mathbf{B} = 0$	Magnetic fluxdensity $\mathbf{B}$ has no sources.

Instead of the capital letters for the vector fields  $\mathbf{H}, \mathbf{B}, \mathbf{E}, \mathbf{D}, \mathbf{J}$  we will also use small letters  $\mathbf{h}, \mathbf{b}, \mathbf{e}, \mathbf{d}, \mathbf{j}$  for them, especially when writing the formulations into the solver input file.

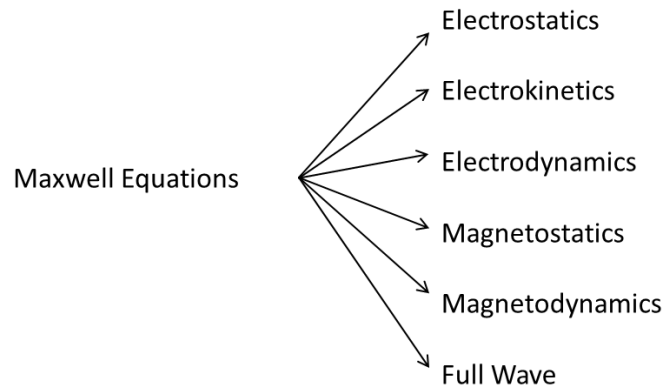
### 1.2.2 Material Equations

In order to properly specify the system (i.e. we have 16 unknowns from the fields and sources but only 7 equations when considering the continuity equation), additional equations are needed: the material equations. By applying material laws magnetic and electrical material properties are included in the analysis. There exist three material laws.





Equation name	Form	Remarks and units
Magnetic relationship	$\mathbf{B} = \mu \cdot \mathbf{H}$	Magnetic permeability $\mu$ is the basic property. $\mu$ is often nonlinear. Then, a $BH$ curve is given. Usually $\mu = \mu_0 \cdot \mu_r$ is used.
Dielectric relationship	$\mathbf{D} = \epsilon \cdot \mathbf{E}$	Electric permittivity $\epsilon$ is the basic property.
Ohm's law	$\mathbf{J} = \sigma \cdot \mathbf{E}$	Electric conductivity $\sigma[S/m]$ is basic property.



### 1.2.3 Electromagnetic Models

The below picture shows use cases of interest in the electromagnetic analysis area which can arise from the Maxwell equations.



The six principle submodels that can be derived from the Maxwell equations particularly differ in the way they account for the effects capacitance, ohm resistance and inductivity. Accordingly, icons for capacitor, resistor and coil can be assigned:

- **Electrostatics**  : here static charges or electrical voltages are the key ingredient and are thus set as sources. As a result of the simulation, we obtain the electric field distribution. This corresponds to a consideration of the capacitive properties (hence the symbol of a capacitor).
- **Electrokinetic (DC Conduction)**  : here, we consider the static distribution of electricity in conductors. The most important feature is the electrical conductivity or the ohm resistance (hence the symbol of resistance).
- **Electrodynamics**  : This is a combination of electrostatics and kinetics. The distribution of the electric field and electric currents in materials (conductors and insulators) are considered. It can also lead to dynamic effects.
- **Magnetostatics**  : We consider the static magnetic field, which can result from permanent magnets and stationary electric currents. As this corresponds to the effect of the inductance, we select the icon of the coil. Read more about this in chapter 'Magnetostatics'.

- **Magnetodynamics** : Result is the magnetic field and eddy currents (Eddy Currents), which result from moving magnets or time-varying currents. A suitable symbol is the coil with ohm resistance, because these two effects are considered.
- **Full Wave (High Frequency)** : This includes considering the full electromagnetic waves. It requires that all three effects of capacitance, resistance and inductance are taken into account. This allows to determine vibrations and resonances, therefore the symbol of the electrical resonant circuit is suitable.

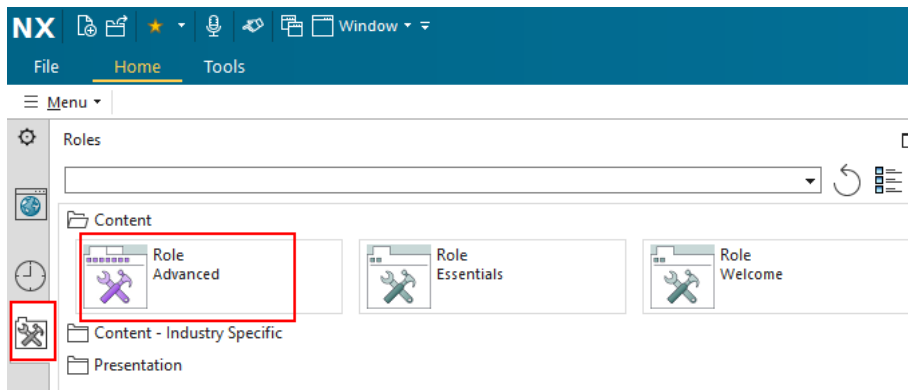
## 2 Recommended System Settings

To simplify the work, we recommend to change the following settings.

### 2.1 Set the Role to Advanced

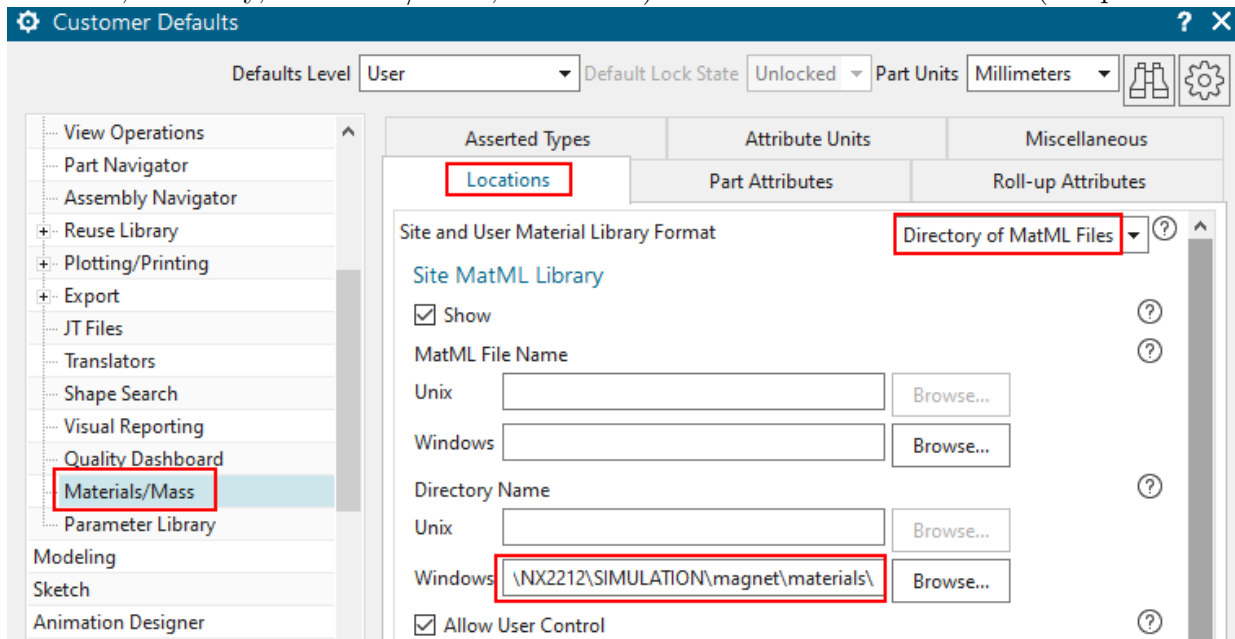
If NX or Simcenter is installed newly, the user-interface shows only the essentials and hides many features. For the simulation applications, it is better to set the role to 'Advanced' to show all necessary buttons. Thus, proceed as follows:

- In the resource bar (left side), click 'Roles',
- open the folder 'Content' and click on the role 'Advanced'.



### 2.2 Set Material Library

For easier finding material properties, we recommend to put a reference to the magnetic material libraries. This must be done in the customer defaults (find it here: File, Utilities, Customer Defaults, Gateway, Materials/Mass, Locations). Set the folders as follows (see picture below):



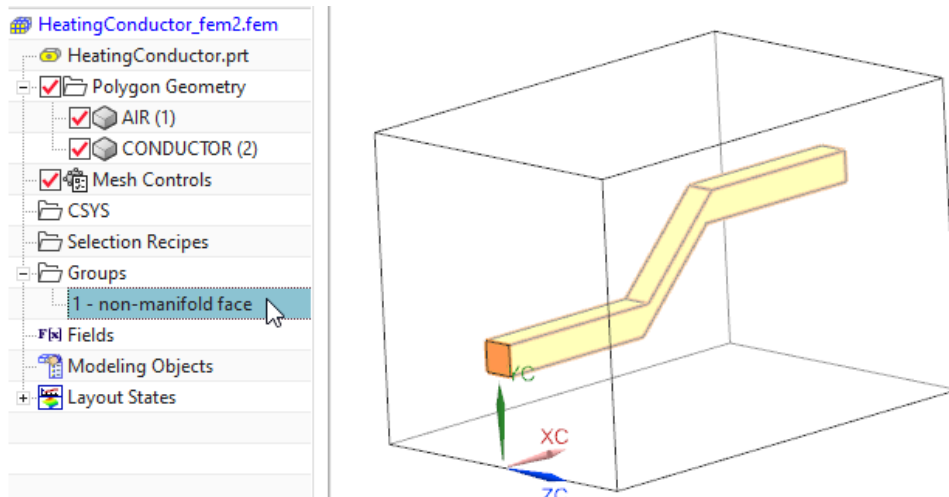
- 'Site and User Material Library Format': 'Directory of MatML Files'

- At 'Site MatML Library', 'Directory Name', 'Windows', key in or browse to the following folder with the magnetic material xml files:

'.../NX< version >/SIMULATION/magnet/materials/'

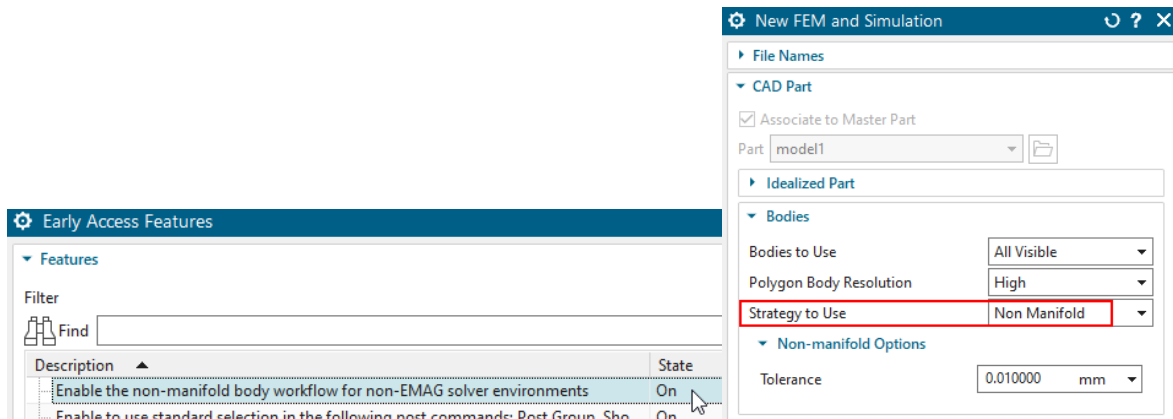
## 2.3 Create Non-Manifold Polygon Bodies

For 3D simulations, there exists a useful feature named 'Non Manifold'. With the use of this, Mesh Mating Conditions (MMC) become obsolete. The system will automatically find all adjacent faces of the solid bodies and match them. More technical spoken: if there are two adjacent faces found, the system will remove one of them and use the remaining face for both solids. Like the MMCs, such matched faces will give a conformal mesh, e.g. the nodes will be identically as we normally need for electromagnetic solutions. For checking, there will be a group named 'non-manifold face' that can be selected to see all the faces that are matched (see picture below). In case of complex geometry, with many small and large faces mixed up, this non-manifold strategy leads to more stable meshing. Thus, we recommend this for all electromagnetic simulations. Also for thermal-flow simulations and in some cases Nastran structurals, this will also be a good choice.



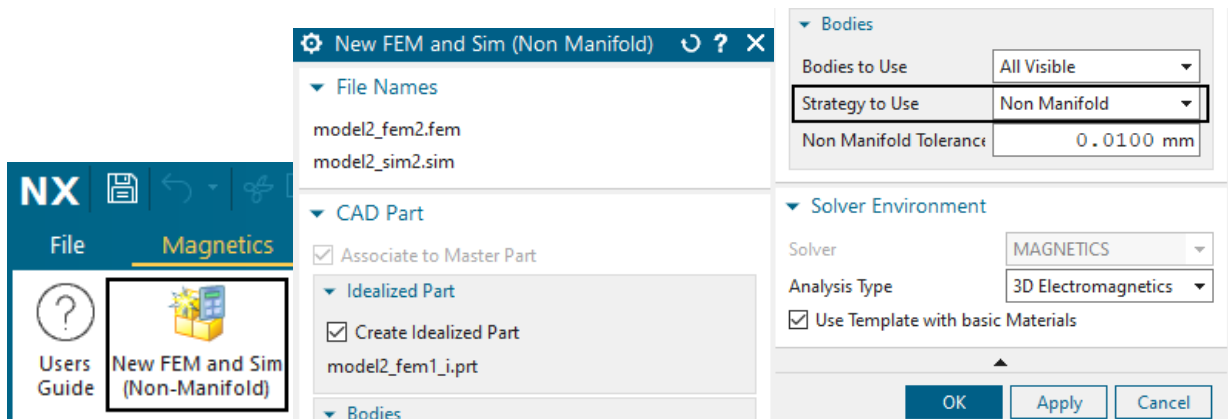
**Activating the Feature** In the NX/Simcenter versions prior to the latest (NX 2406), this feature must be activated in different ways (unfortunately), as we describe in the following.

- In version **2406**, there is no need to activate the feature. It is visible by default in the dialogue 'New FEM and Simulation'.
- In version **2312**, it is activated by two steps: First, we enable an early access feature that we find here: File, Utilities, Early Access Feature. Search for the feature 'Enable the non-manifold body workflow ...' and set this to the state 'On'. Then, when creating a new Fem file, there will be the 'Strategy to Use' option that we set to 'Non Manifold'.



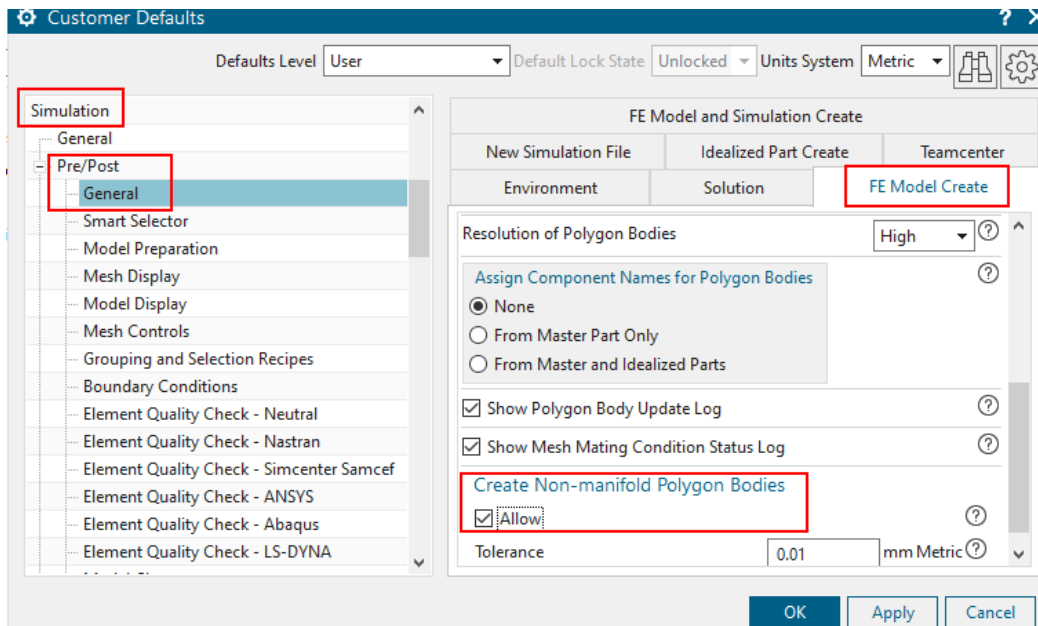
Alternatively, it can be activated in the same way as in versions 2212 and 2306 (see below).

- In versions **2212** and **2306**, it is activated as follows:  
Use the button 'New FEM and Sim (Non-Manifold)' from the Magnetics toolbar.



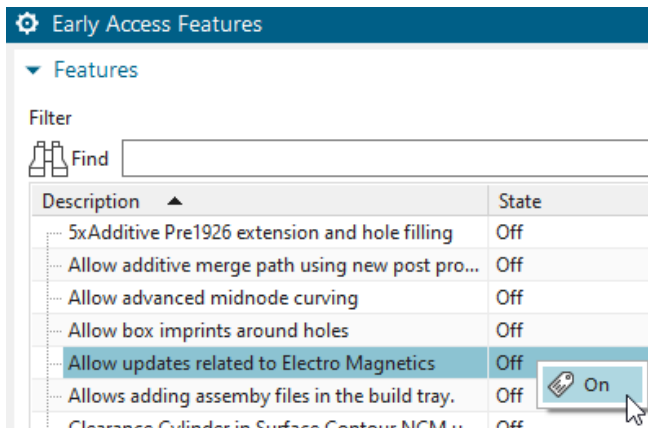
A dialogue will appear that looks similar to the conventional one, but it contains the option 'Strategy to Use' that is set to 'Non Manifold' by default. When clicking OK the system will create a Fem and Sim file with activated non-manifold feature. Additionally, the files are created using the MAGNETICS template files if the option 'use Template with basic Materials' is active. Thus, there will be some basic materials available in the FEM part. When Teamcenter is used, this way is not possible and the following alternative must be used.

- Alternatively, in versions **2212**, **2306**: when creating the FEM file, set the solver first to 'Simcenter MAGNET' and create the FEM file. Later, edit the FEM file and set the solver to 'MAGNETICS'. The non-manifold feature is active then. This way also works with Teamcenter.
- In the versions **2007 (2022.1)** and **2206 (2022.2)**, this feature is named 'Create Non-manifold Polygon Bodies' and it is found in the customer defaults. (find it here, see picture below: File, Utilities, Customer Defaults, Simulation, Pre/Post, General, FE Model Create, Create Non-manifold Polygon Bodies, Allow)



If a FEM file is created, it will have this feature active.

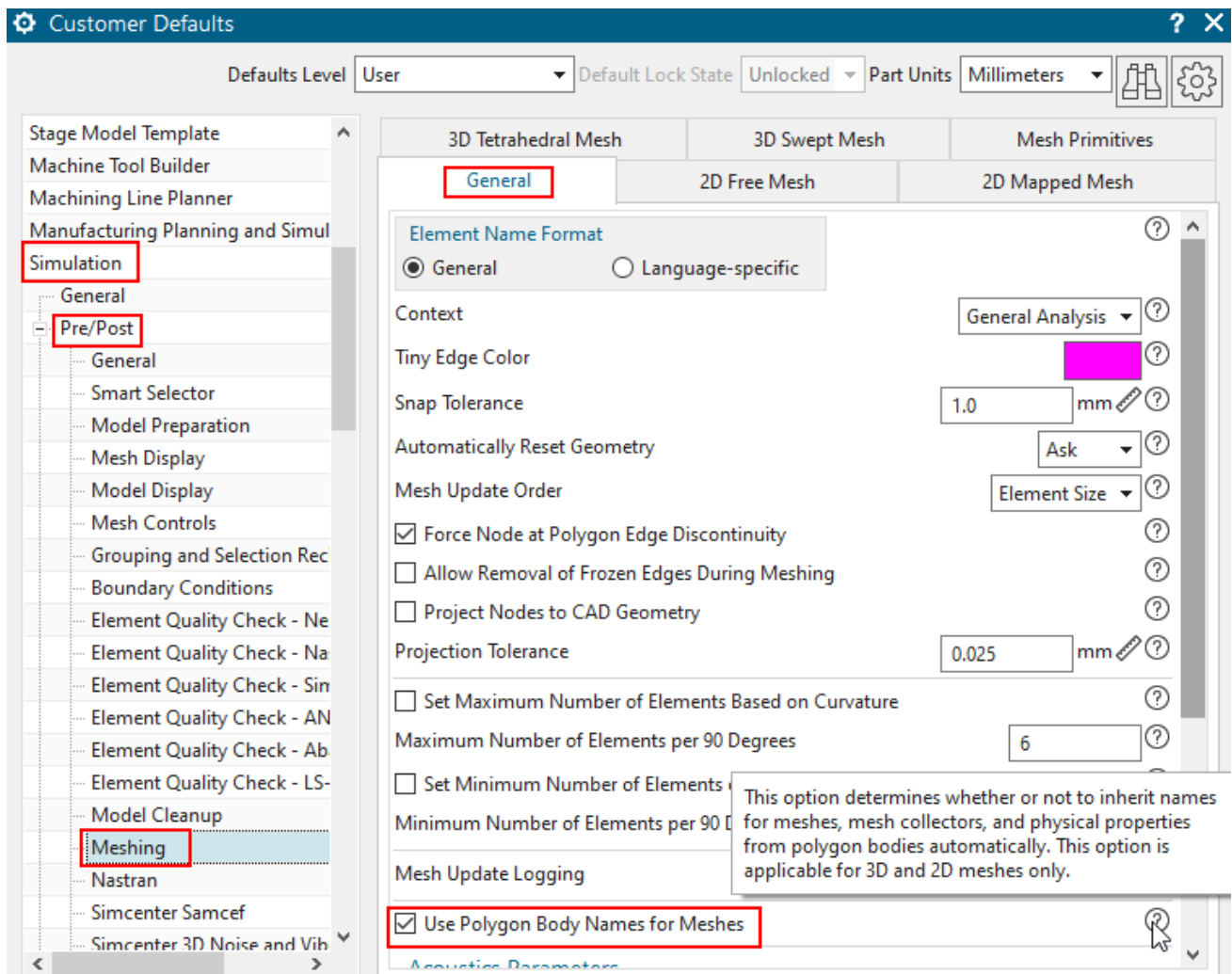
- In version **1953 (2021.1)**: find it here, see picture below: File, Utilities, Early Access Feature), named 'Allow updates related to Electro Magnetics'. Set this 'On'.



If now a FEM file is created, it will have the feature active.

## 2.4 Use Polygon Body Names for Meshes and Physicals

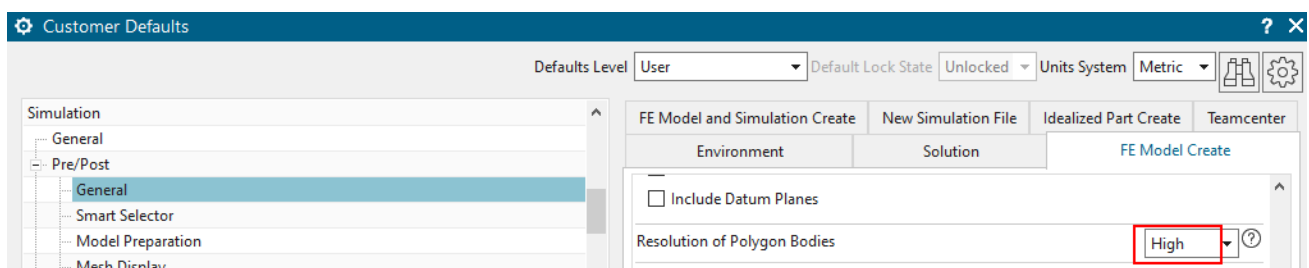
If the NX/Simcenter version is 1899 (2020.1) or higher, we recommend to enable the following setting in the customer defaults (find it here: File, Utilities, Customer Defaults. Then: Simulation, Pre-Post, Meshing, General) named 'Use polygon body names for 3D meshes'.



When this setting is active, the names of the polygon bodies will be transferred to meshes, mesh collectors and physical property tables. This simplifies the work and therefore, we recommend this setting.

## 2.5 Use High Resolution for Polygon Bodies

We recommend to set the standard resolution of polygon bodies to 'High' because this often simplifies meshing of complex geometries.(find it here: File, Utilities, Customer Defaults. Then: Simulation, Pre-Post, General, FE Model Create)



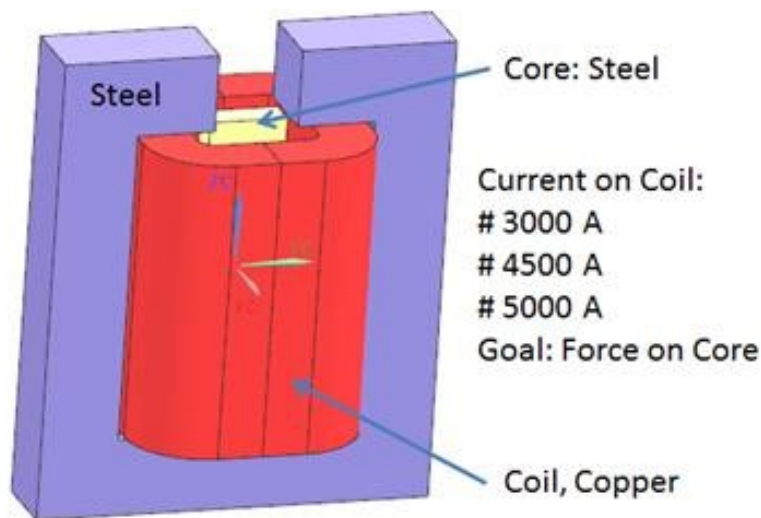
## 3 CAD Basics

### 3.1 3D Solenoid (Beginner)

Goal of this tutorial is to learn some basics of CAD modeling in NX or Simcenter starting from scratch. This tutorial shows very typical CAD techniques: sketching, extruding and subtracting. If the user is familiar with these he will be able to model much more complex geometry in a very similar way too.



For those who directly want to see the result: The completed CAD file for the Solenoid example can be found in this folder:

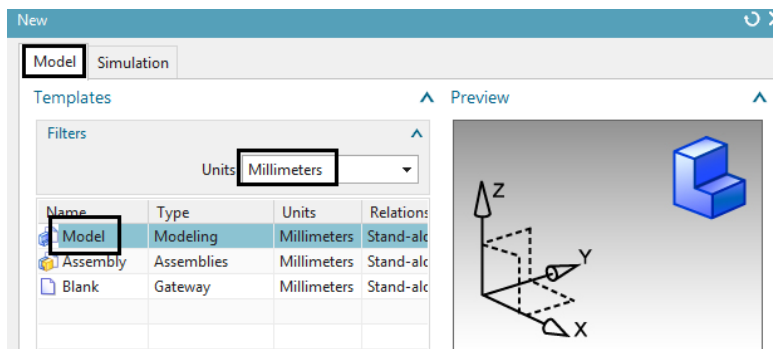
'Your Installation'/MAGNETICS/docu/Tutorials/0.CADbasics/0.1Team20/complete If desired, the CAD file resulting from this tutorial can be used for starting the simulation tutorial 3D Simulation: 'Static Force on Solenoid'

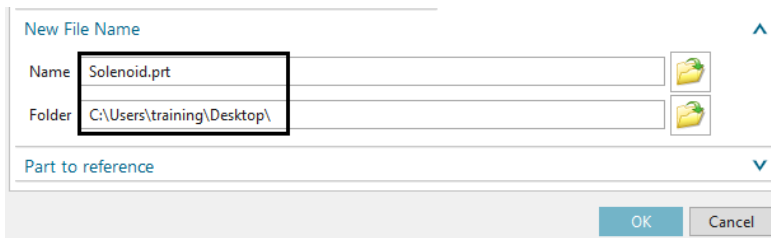


Estimated time: 45 min. Follow the steps in Simcenter:

#### 3.1.1 Preparations

1. Start the Program Simcenter  (or NX). Use Version 10 or higher.
2. In Simcenter click 'New' ,
  - Click on register 'Model',
  - in box 'Templates' set the 'Units' to 'Millimeters' and select 'Model',






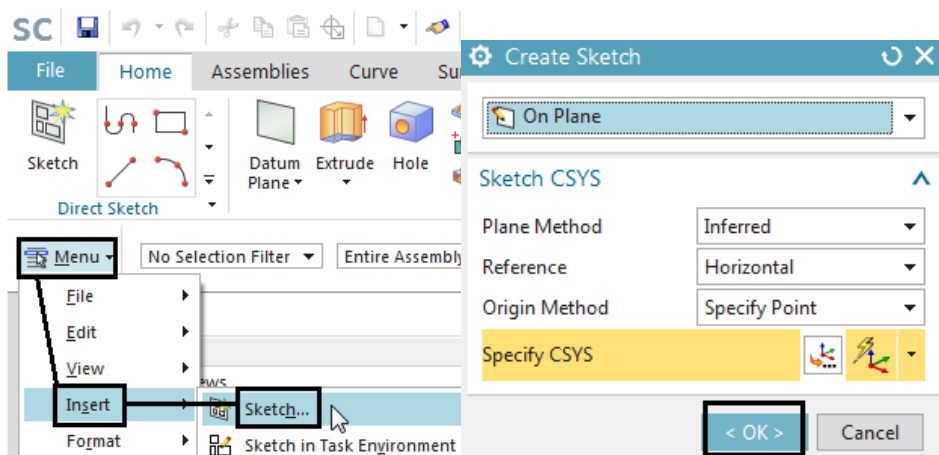
- in box 'New File Name': for 'Name' key in 'Solenoid.prt'.
- for 'Folder' navigate to any folder you want.
- Click OK. The empty part file is created and the system automatically switches into the 'Modeling' application.


### 3.1.2 Sketch the Core

The core is a simple block positioned in the middle. We could use the 'Block' feature for that. But we want to use a more general way: First sketching a rectangle and then extruding it to a block. Sketching means we create curves on a two dimensional plane. Following such curves are extruded or revolved and thus 3D geometry results from the sketch curves. Such features 'Sketch' and 'Extrude' are very comfortably and powerful. Many geometry types can be created by this way. Following we start CAD features by using the 'Menu' command (see picture below). By doing so such features are found always at the same location. Of course most feature buttons can also be found somewhere in the toolbar where is may be faster to click. But we prefer the 'Menu' command for this tutorial because the toolbar locations may change from user to user.

1. Click on 'Menu' → Insert → 'Sketch' 

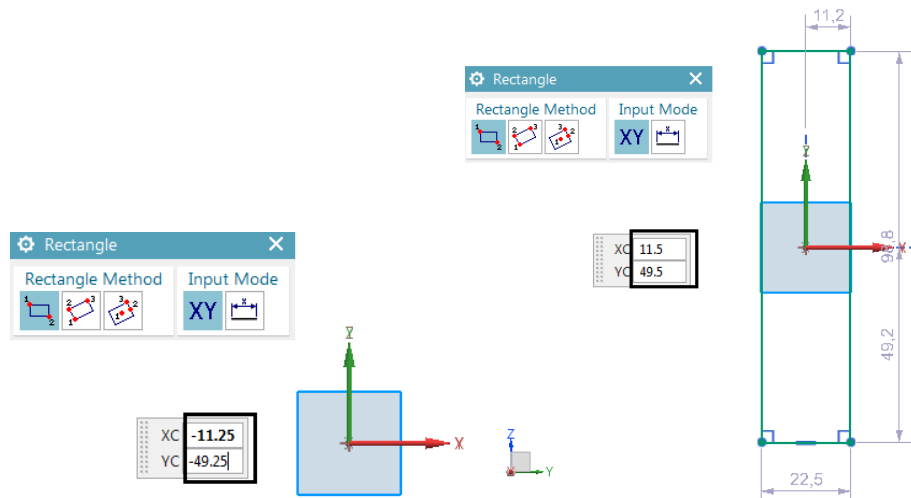
2. A window 'Create Sketch' appears asking for a plane to sketch on. Select the Y-Z (in absolute coordinates) plane as shown in the picture below right and click OK. The sketch environment is started.




3. Click on 'Rectangle'  or simply key in 'r' to use the shortcut. A window 'Rectangle' appears asking for the definition of the rectangle.


- key in the first coordinate (-11.25 -49.25 mm) into the little entry field and press 'Return'.

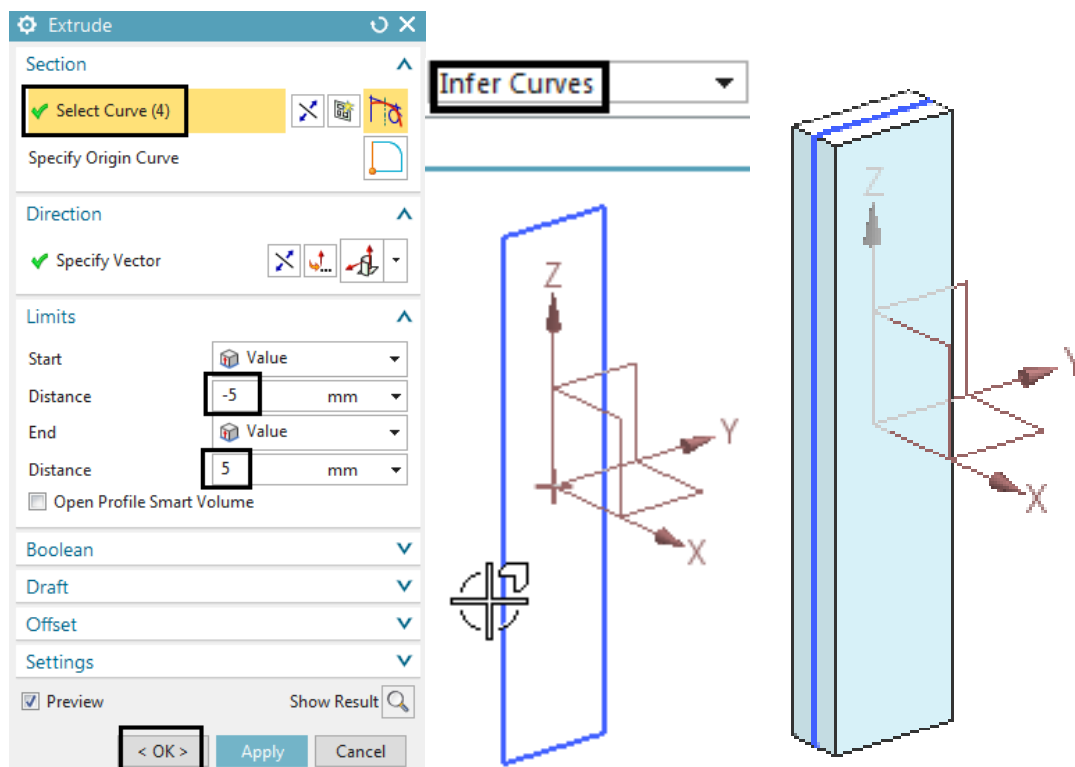
- key in the second coordinate (11.25 49.25 mm) into the entry field and press 'Return'. The rectangle is created.



- Press 'Escape' to cancel the dialog.
- Click on 'Finish Sketch'  or use the shortcut 'Ctrl+Q' to cancel the sketch environment.

### 3.1.3 Extrude the Core

1. Click on 'Menu' → Insert → Design Feature → 'Extrude'  or simply key in 'X' to use the shortcut. The 'Extrude' dialog appears.
2. Set the selection type to 'Infer' and select the sketch.



3. Key in the two Distance values (-5 and 5 mm) and click on OK. The block is created.


### 3.1.4 Zoom, Rotate and Pan

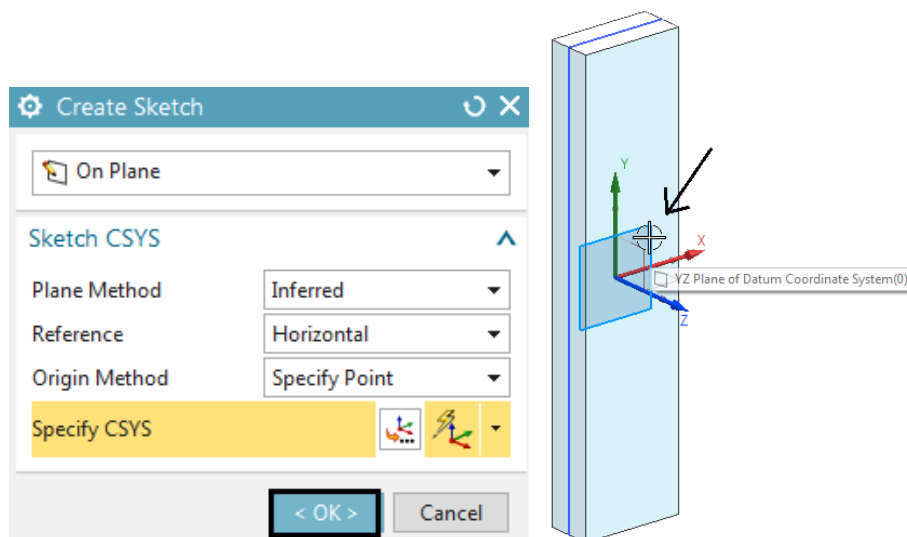
Of course it is necessary to know how to use zoom, rotate and other display options. Following is a list of useful options. Feel free testing them now:


- Rotate: Click and hold the middle mouse button (MMB). Then move the mouse in the graphics area. The block will rotate.
- Pan: Click and hold the middle and the right mouse button together. Again move the mouse. The block will move (pan) without rotation.
- Zoom in/out: Scroll the mouse wheel to zoom in or out. Alternatively click and hold the middle and the left mouse button together.
- Fit: To fit all visible geometry into the graphics area use the shortcut 'Ctrl+F'. Alternatively right click in the graphics area and from the appearing window click on 'Fit'.
- Switch between shaded and wire frame display: right click in the graphics area and from the appearing window click on 'Rendering Style'. Choose from the options there: For instance 'Static Wireframe' or 'Shaded'.

### 3.1.5 Sketch the Iron

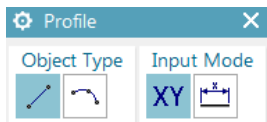
Again sketching is a convenient way for modeling the iron.


1. Start the sketcher by clicking on 'Menu' → Insert → 'Sketch' 
2. The window 'Create Sketch' appears asking for the plane to sketch on. Select the same plane as already for the core. Click OK.



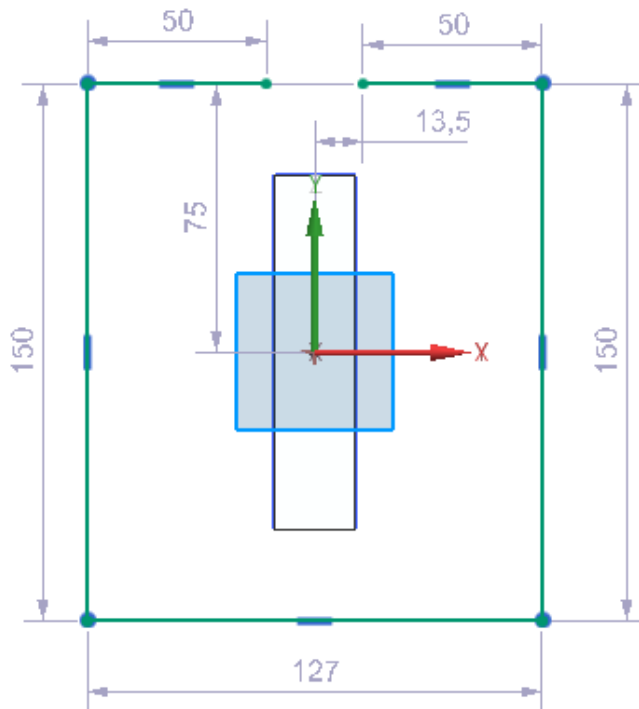
3. Following we are creating several line segments. Therefore the simplest way is to use the 'Profile' method. Click on the 'Profile'  button or simply key in 'Z' to use the shortcut.


A window asking for coordinates appears.



4. Key in the following x,y coordinates into the little entry field. To begin a new x,y entry click . Press 'Tab' to switch between x and y and press 'Return' after each coordinate pair. Use these x,y value pairs:


- -13.5 75 , -63.5 75 , -63.5 -75 , 63.5 -75 , 63.5 75 , 13.5 75
- There appears a profile as shown below.



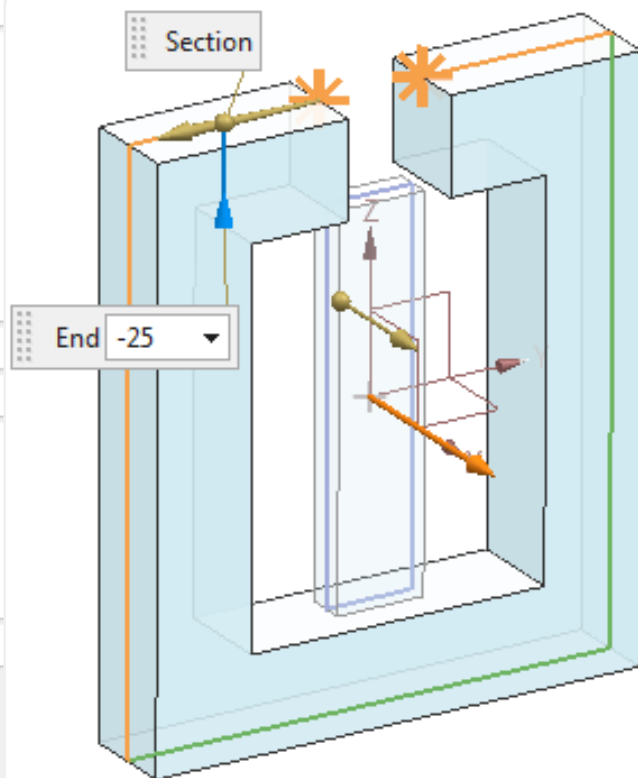
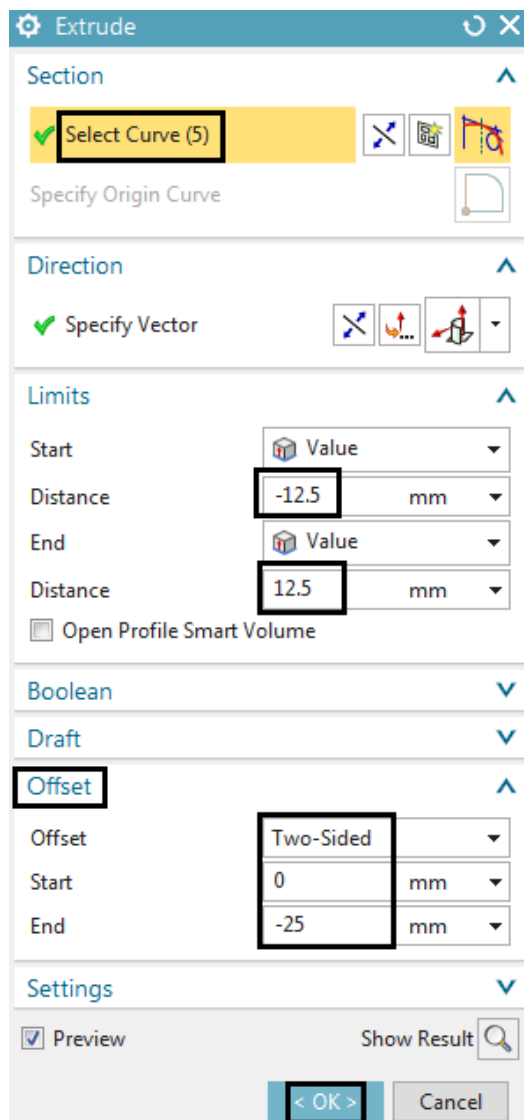
- Press 'Escape' to cancel the dialog.
- Finish the sketch 

### 3.1.6 Extrude the Iron

The following extrude feature is a little special because we use it with an 'Offset' option. This option allows creating a solid body from a (non closed) profile. In many cases this is a convenient way to create geometry.

1. Click on 'Menu' → Insert → Design Feature → 'Extrude' . The 'Extrude' dialog appears.
2. Set the selection type to 'Infer' and select the new sketch.

3. Key in the two Distance values (-12.5 and 12.5 mm) and check the direction of the specified blue vector (your iron geometry should look like the one in the picture below).
4. Click on the box 'Offset' to make such options visible.
5. Set the 'Offset' to 'Two Sided' and key in 0 and -25 mm in the field. (Attention: Sometimes you may need the opposite: 25 instead of -25. Please check your iron geometry against the one in the below picture)
6. Click on OK. The iron geometry is created.



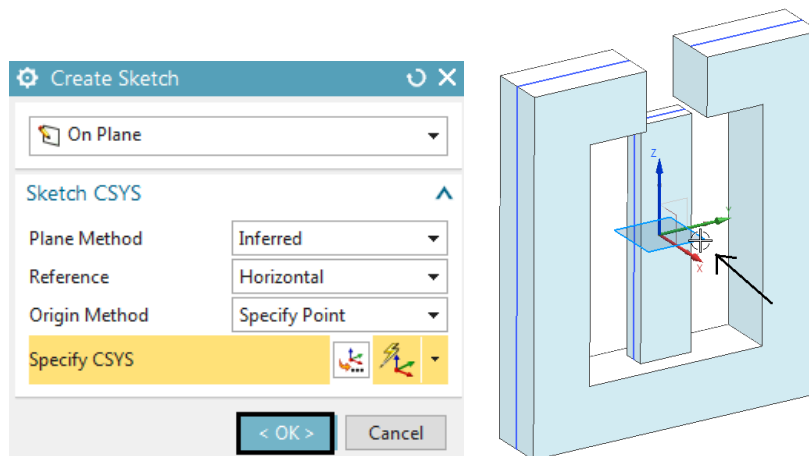
### 3.1.7 Sketch the Coil


For the coil we also use the sketch/extrude method and we also exploit the 'Offset' option. It can be seen that these features are really useful for many types of geometry.


1. Start the sketcher by clicking on 'Menu' → Insert → 'Sketch'

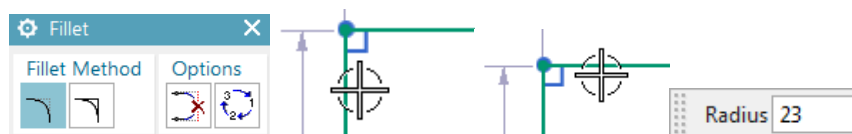


2. The window 'Create Sketch' appears. Now for the coil, select the plane (X-Y) that is perpendicular to the core and iron plane.

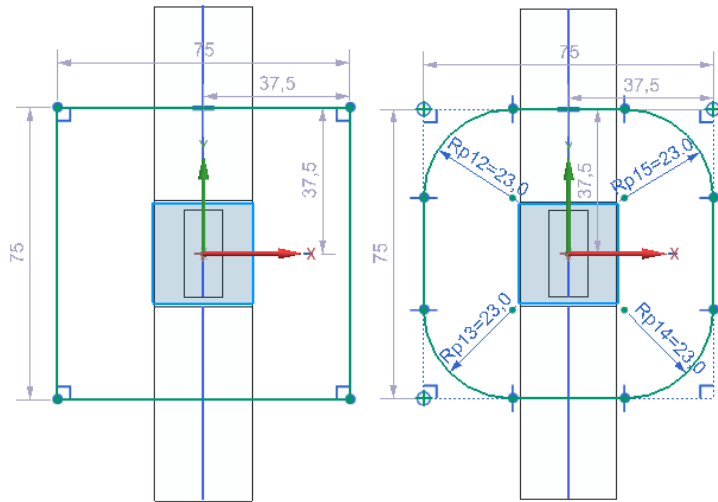


3. Following we are sketching a rectangle first and then use the fillet function to create circles at the four corners.
4. Click on 'Rectangle'  or simply key in 'R' to use the shortcut. The window 'Rectangle' appears asking for the definition of the rectangle.
5. Key in the first coordinate (37.5 37.5 mm) into the little entry field and press 'Return'.
6. Key in the second coordinate (-37.5 -37.5 mm) into the entry field and press 'Return'. The rectangle is created. See picture below left. Press 'Escape' to cancel the dialog.

7. Click on 'Fillet'  or simply key in 'F' to use the shortcut. A window 'Fillet' appears asking for line pairs.
8. Select the two lines of a corner and key in the radius 23 mm. Press 'Enter' and the fillet is created.



9. Repeat the fillet creation for all four corners of the rectangle. The result is shown in the below picture right.




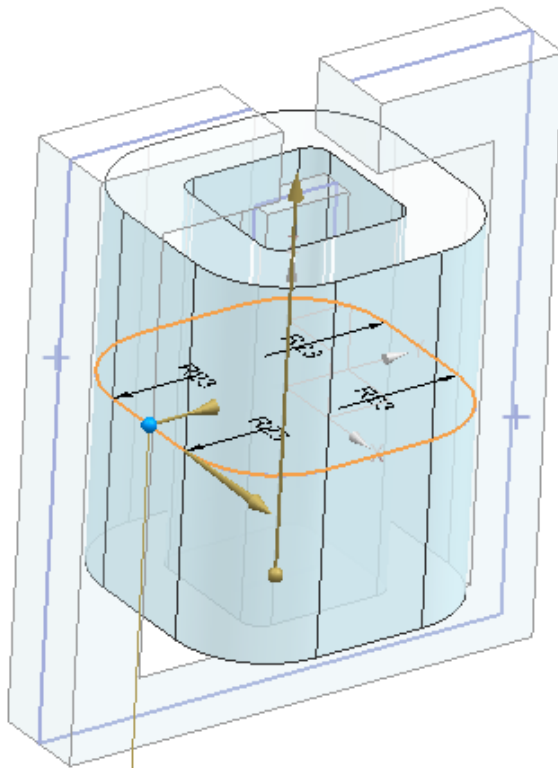
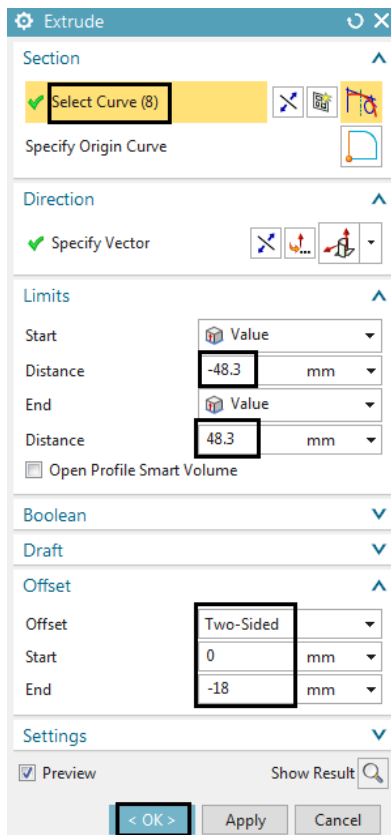
10. Finish the sketch



### 3.1.8 Extrude the Coil

For extruding the coil we again use the 'Offset' option.

1. Click on 'Menu'→Insert→Design Feature→'Extrude' . The 'Extrude' dialog appears.




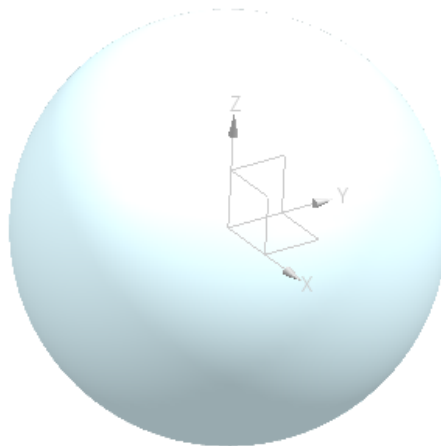
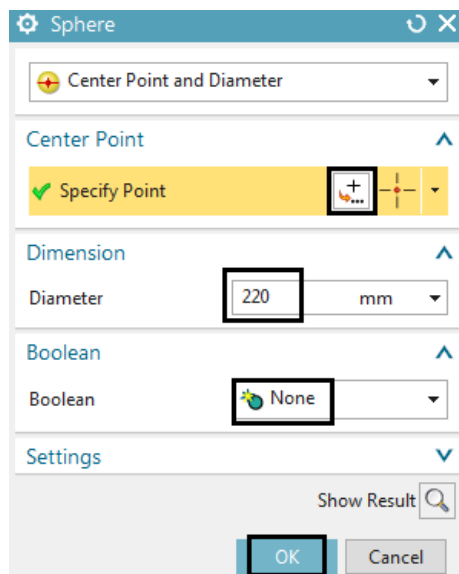
2. Set the selection type to 'Infer' and select the coil sketch.
3. Key in the two Distance values (-48.3 and 48.3 mm).


4. Click on the box 'Offset' to make such options visible.
5. Set the 'Offset' to 'Two Sided' and key in 0 and -18 mm in the field. (Hint: In some cases you will need 18 instead of -18.)
6. Click on OK. The coil geometry is created.
7. Later in the simulation we will define winding directions for this coil. There it is necessary having one joined curve instead of several pieces. Therefore proceed as follows: Click on Menu→Insert→Associative Copy→'Extract Geometry'. The 'Extract Geometry' dialog appears. Check the type is set to 'Composite Curve'. Select the 8 sketch curves and click OK. The new curve is created.

### 3.1.9 Create a Spherical Air Volume

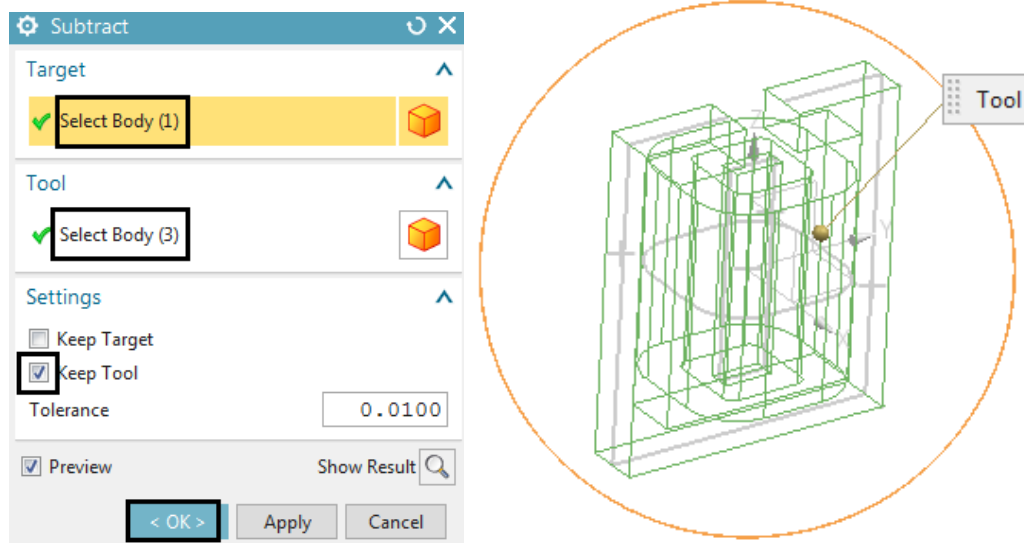
In Electromagnetics and also in Flow simulations it is necessary to model the air or fluid volume. In many cases this can be a simple sphere or a box. The detailed air volume for later meshing results from a subtraction of all solenoid volumes from that air volume. For easier meshing we will finally subdivide the face of the sphere. We model such a sphere in the following.


1. Click on 'Menu' → Insert → Design Feature → 'Sphere' . The 'Sphere' dialog appears.
2. Click on the 'Point Dialog' and key in the center point coordinate (0 0 0).
3. Key in 220 mm at 'Diameter' and click OK. The sphere is created.

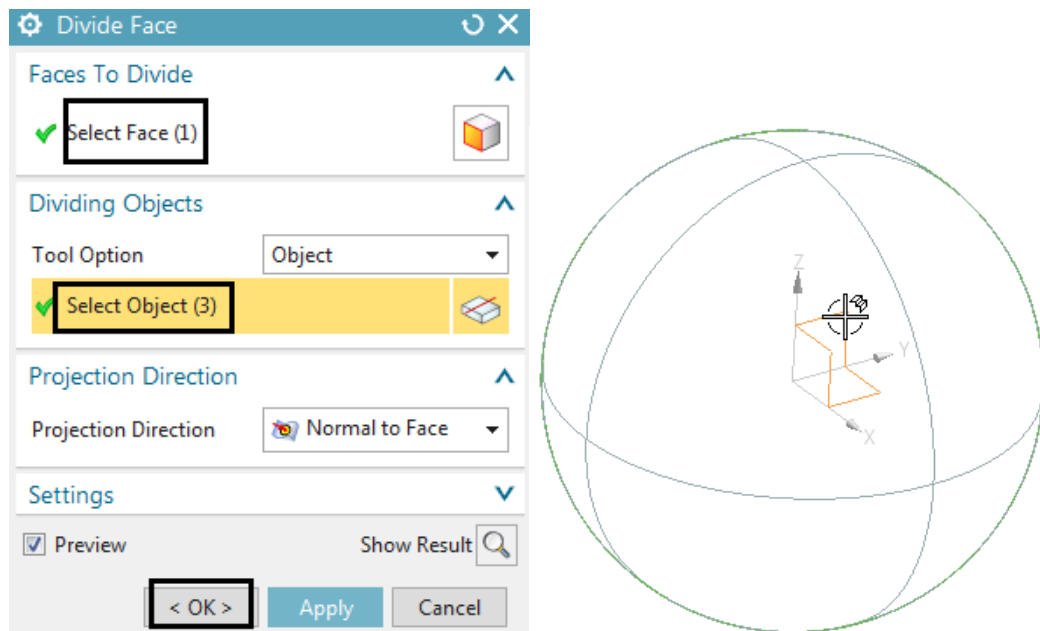


4. Click on 'Menu' → Insert → Combine → 'Subtract' . The 'Subtract' dialog appears.
5. Click on 'Target - Select Body' and select the sphere volume.
6. Click on 'Tool - Select Body' and drag a window over everything. The three solenoid bodies are selected.

7. Activate the button 'Keep Tool' and click OK. The operation is performed.



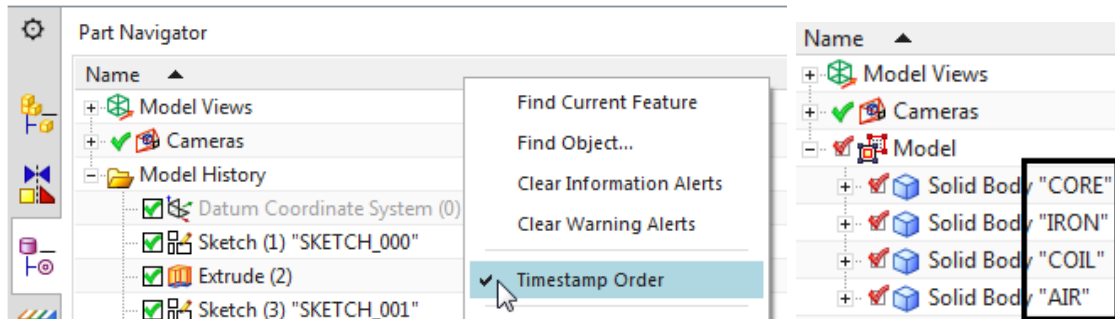
8. Click on 'Menu' → Insert → Trim → 'Divide Face' . The 'Divide Face' dialog appears.
9. Click on 'Faces to Divide - Select Face', set the selection filter to 'Single Face' and select the spherical face.
10. Click on 'Dividing Objects - Select Object' and select the three planes of the datum coordinate system.
11. Click OK and the face subdivisions are performed.



### 3.1.10 Assign Names to the Bodies

Later in the simulation working with speaking names is quite helpful. Therefore we want to assign names to the bodies. Such bodies will result in so called 'Polygon Bodies' later and these will be used for creating finite element meshes.

1. In the 'Part Navigator' right click on the grey border (see picture below) and deactivate 'Time Stamp Order'. Now bodies are shown in the navigator instead of the creation order of CAD features.
2. Use RMB on each of them, chose 'Rename' and assign speaking names.



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

## 4 Magnetostatic

Magnetostatics is used when information of the magnetic field that surrounds a conductor is needed. In this context, a *DC Conduction Steady State* analysis is sometimes used as a preprocessing step, and the resulting currents are used as input to a subsequent magnetostatics analysis. This would be the case, for example, when analysing electromagnets. The fundamental material property for performing magnetostatics analysis' is the relative magnetic permeability  $\mu_r$ . For nonlinear magnetostatics analyses, a more general material relationship may be needed, such as a functional relationship between the magnetic field and the magnetic flux density: a so-called B-H curve. The ultimate goal of a magnetostatics analysis is, in many cases, to compute the forces and torques in a system of magnetic components.

Analysis of permanent magnets constitutes an important, special case of magnetostatics analysis. In this case, a permanent magnetization is the source of the magnetic field instead of an electrical current. In such cases, the magnetic flux strength and direction as well as forces are important analysis results.

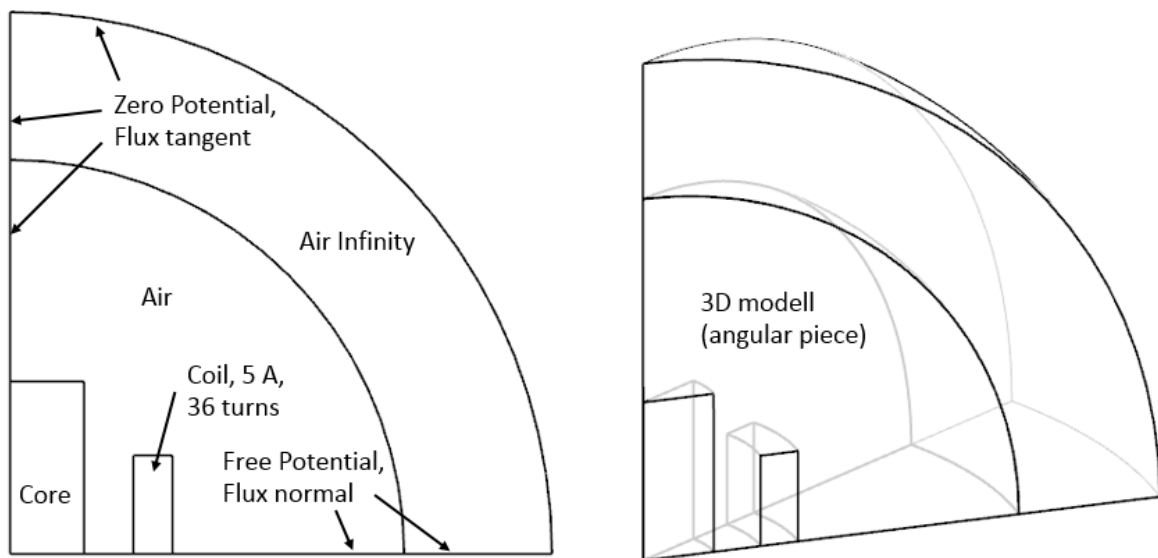
## 4.1 Core and Inductor (Beginner)

In this example it is shown how to build up and analyse simple models in 2D or axial-symmetric situations. An electric coil is positioned near a magnetic core. The electric current in the coil creates a rotating magnetic field (ampere law), that affects the magnetic core material. We want to calculate for the magnetic field strength and flux density in the core. Also, we want to find the mechanical forces acting on the core.

The picture below left shows a sketch representing the model. In the 2D case, we assume an extruded geometry that is simulated by flat 2D elements and an applied thickness (150 mm.) On the right side, there is shown how this would look like in an axial-symmetric case (an angular piece is shown).




The coil is loaded with electric current of 5 Ampere DC and 36 winding-turns.

Following to this 2D analysis (with thickness) we will use the same model to analyse for the axial-symmetric situation.

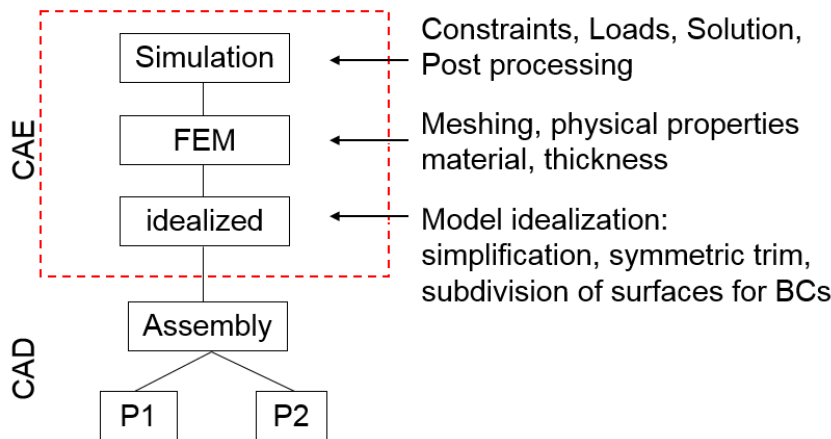


Estimated time: 45 min. Follow the steps in Simcenter:

### 4.1.1 Preparations


1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/3.MagSta/3.1CoreInductor.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program Simcenter  (or NX). Use Version 10 or higher.
4. In Simcenter, click Open  and navigate to folder 'start'. Select the file 'CoreInductor.prt' and click OK. (Maybe you must set the file filter to 'prt')
5. From toolbar Application click on 'Pre/Post'  and the system switches into the simulation application.

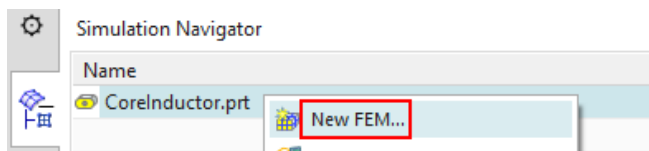
6. Hints: All information describing the Simcenter simulation is managed in basically three files (picture below): The Sim, the Fem and the Idealized file. The Sim file contains the solution, boundary conditions, loads. The Fem file contains the finite-element meshes, material properties and other physical properties. The Idealized file is optionally and can be used for simplifications or modifications of the CAD geometry without changing the original CAD file.



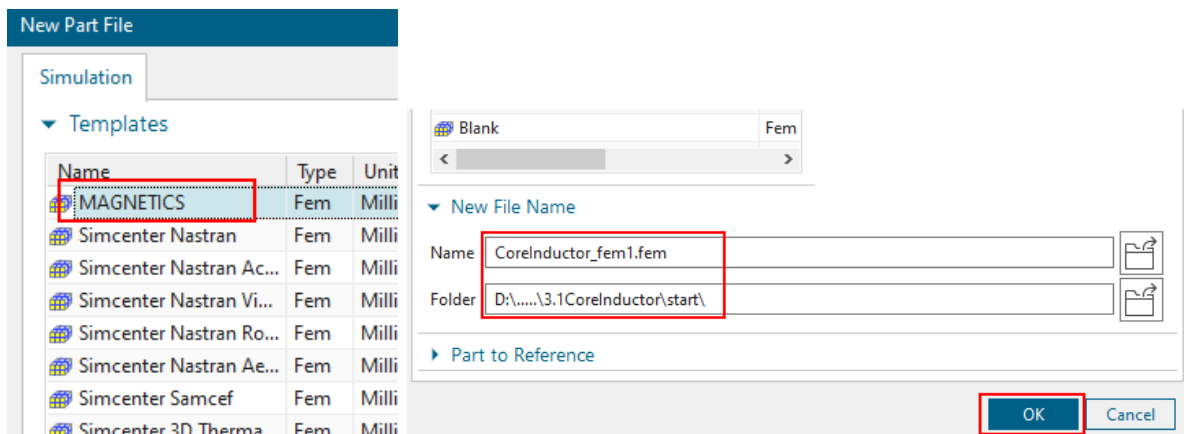
#### 4.1.2 Fem and Sim File Creation

Following, we will first create the Fem and Sim file structure and then we will fill it up with data. We will create the Fem and Sim files in two separate steps, which has the advantage that we can choose a Fem-file-template. The template already contains basic material properties. The 'Simulation Navigator' will allow us navigating in the file-structure.

1. Click on 'New FEM' .  
( 'New FEM' is also found at RMB on the master part in Simulation-Navigator)

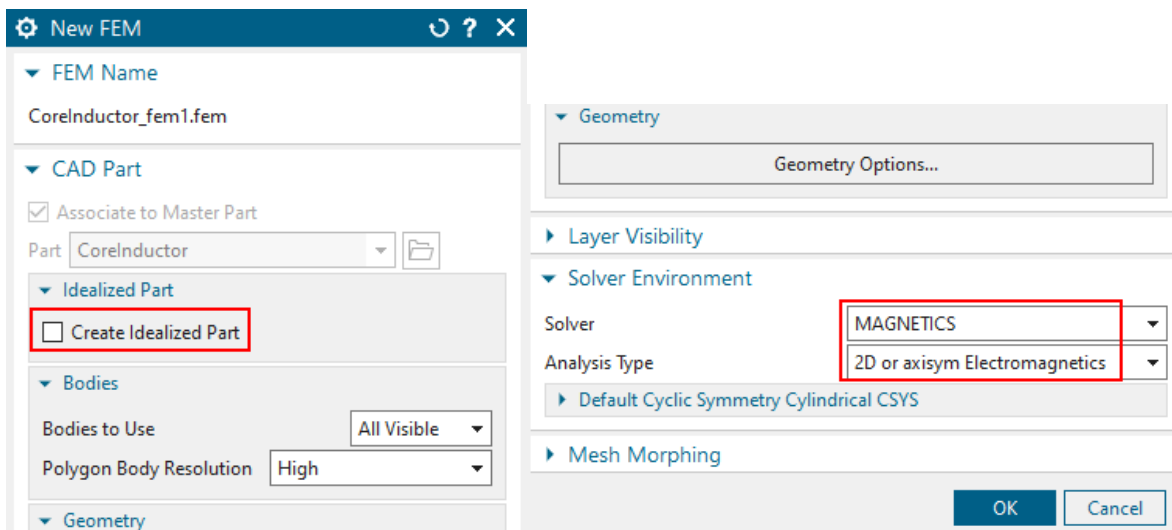


2. In window 'New Part File', select the MAGNETICS-template, check file name and folder (use work part location for all parts) and click 'OK'.



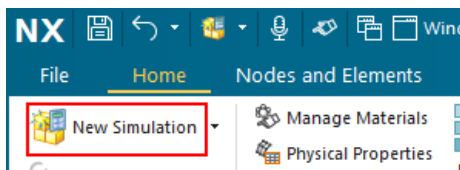
(Messages 'Unsecure Solver Language File' may appear and can be accepted)

3. The window 'New FEM' appears.

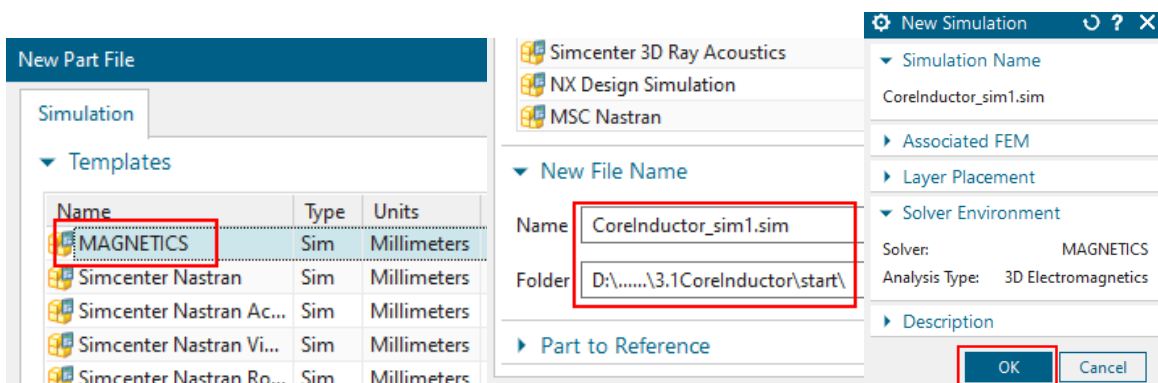


- Switch off 'Create Idealized Part' because the geometry is already very simple. Thus, we will have one file less and the file structures is simpler.
- Set the Solver to MAGNETICS and Analysis Type '2D or axisym Electromagnetics' and click OK. The Fem file is created.

4. Click on 'New Simulation'  (or find this RMB on the Fem file).




5. A window 'New Part File' appears in which you again select the MAGNETICS-template and check for the new name and folder. Click 'OK'. In a further window 'New Simulation', also click 'OK'.



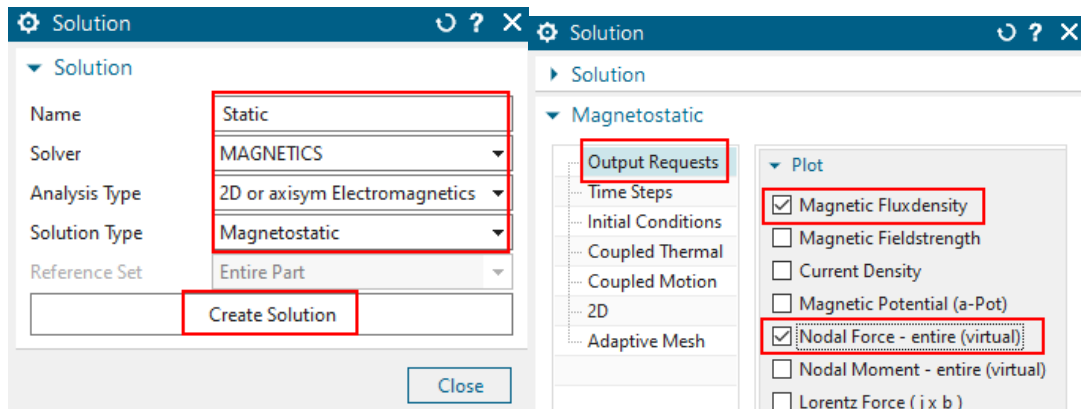
The Sim file is now also created.

6. Following, a window 'Solution' appears

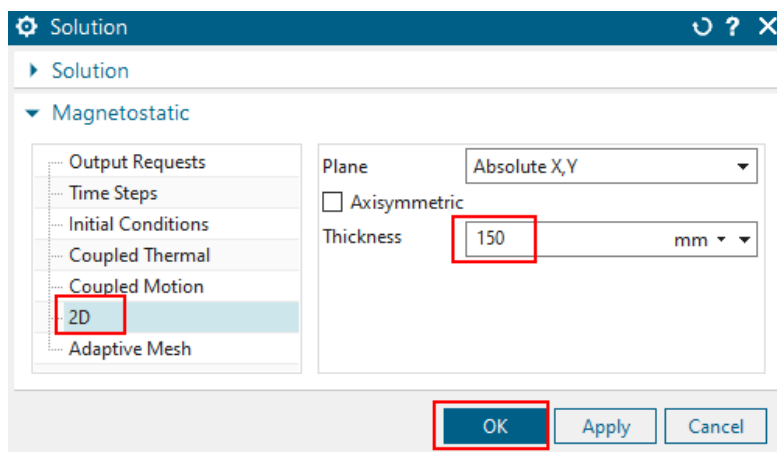
(if this does not automatically appear, click on 'Create Solution' )

- assign the Name 'Static', accept the default Solution Type 'Magnetostatic'. Click 'Create Solution' and

- in register 'Output Requests', activate the results 'Magnetic Fluxdensity' and 'Nodal Force - entire (virtual)'.



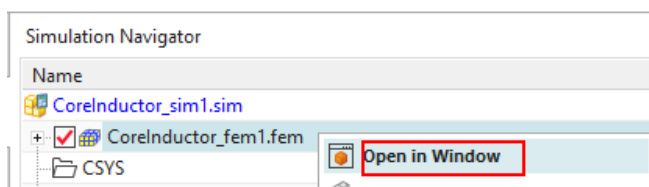
- Click on register '2D' and set the 'Thickness' to 150 mm.



- All other settings are well predefined. Click OK. The solution will be created and the system will jump to the Fem file.

### 4.1.3 Fem File Steps

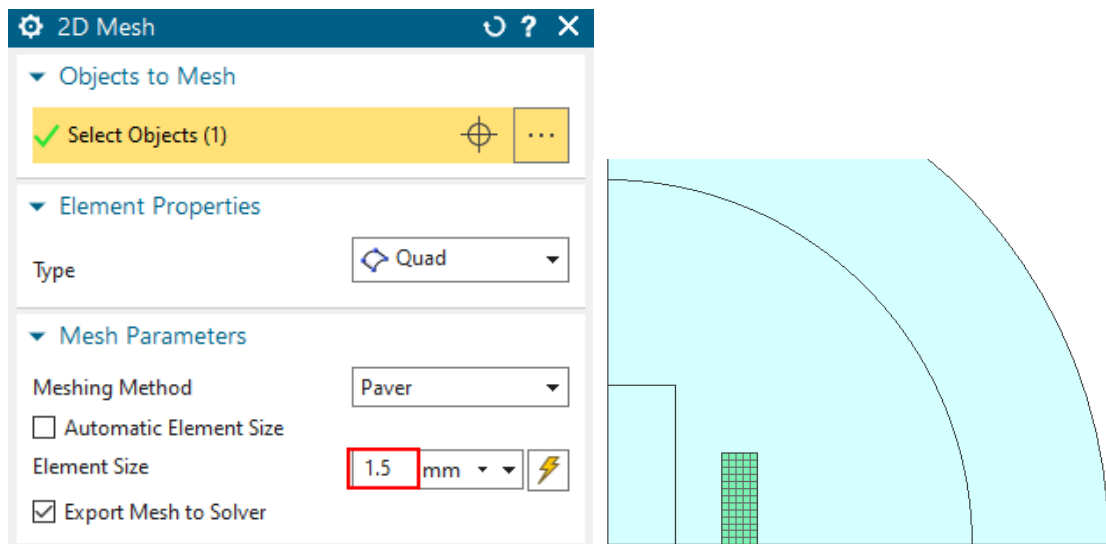
1. We are now going to create finite element meshes. Therefore, use the Simulation Navigator to set the Fem file to 'Open in Window'.




2. Mesh the Coil

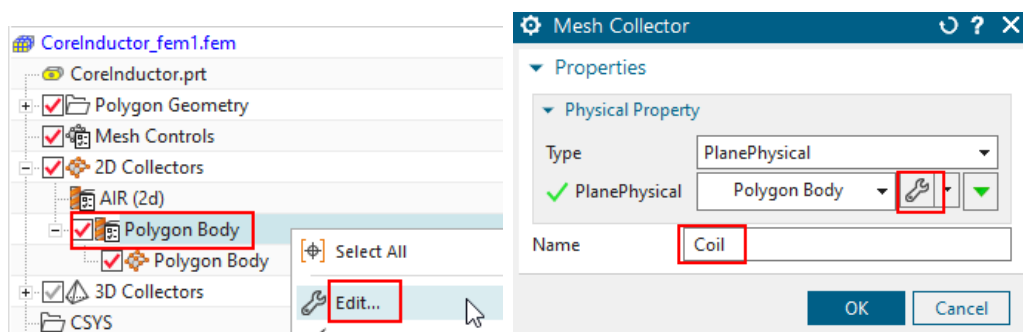


- Click on '2D Mesh' from toolbar Home→Mesh and select the the coil face.
- Set the Type to 'Quad' and the element size to 1.5 mm.  
Hint: Tri elements or other sizes would also work.
- Click OK and the mesh is created.

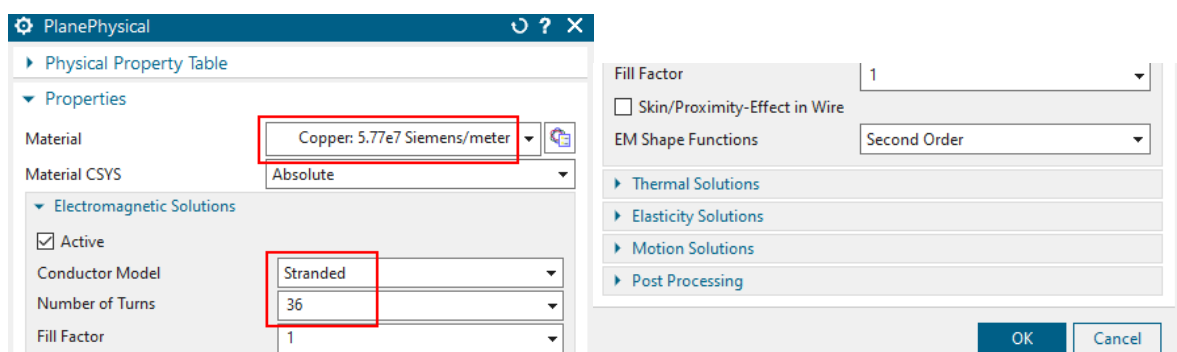



### 3. Assign Properties to the Coil

- Double click on the newly created 2D Collector.
- In the following window, set the Name to 'Coil' and click 'Edit' .  
Hint: It is recommended to assign meaningful names because those names will appear in the results and legends of all following result tables.



- A new window 'PlanePhysical' appears.




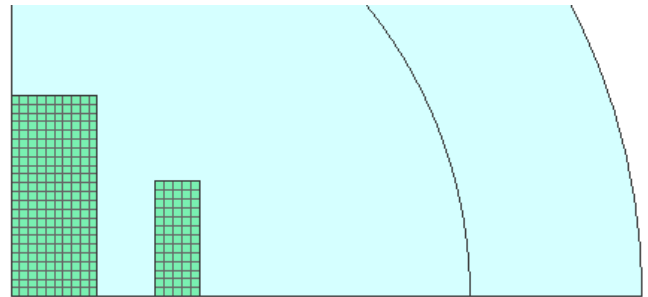
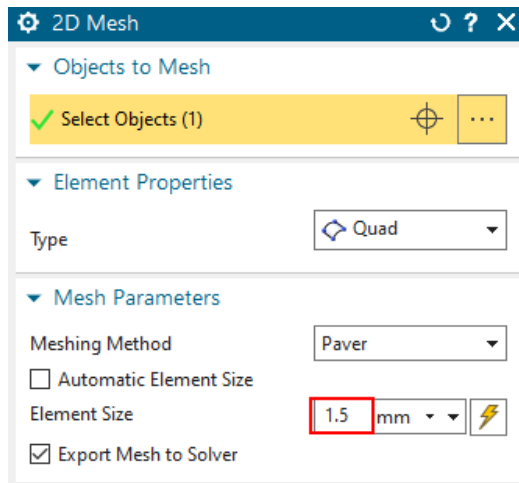
- At 'Material', choose 'Copper...', from the list.  
Hint: Because we have used the Fem file template, there are already some basic materials in the list. For more flexibility, choose from many materials from libraries or define your individual material properties by clicking 'Choose Material' .
- set the 'Conductor Model' to 'Stranded' and apply the value 36 for 'Number of Turns'. Accept the default 1 for 'Fillfactor'.

Hint: The coil fill factor represents the fraction of the coil cross section that the conductor occupies. You can use this factor to adjust coil resistance.


- Click OK, OK to finish the definition of the coil.

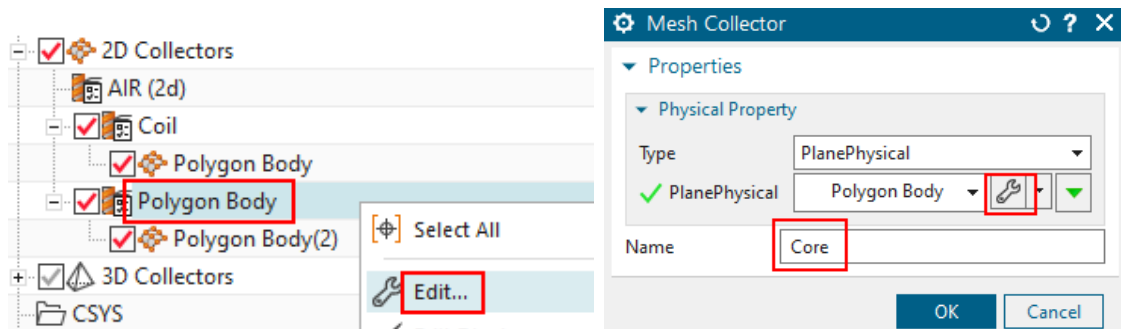
#### 4. Mesh the core:

- Click on '2D Mesh'  and select the core face.
- Use quad elements of 1.5 mm size. Click OK and the mesh is created.

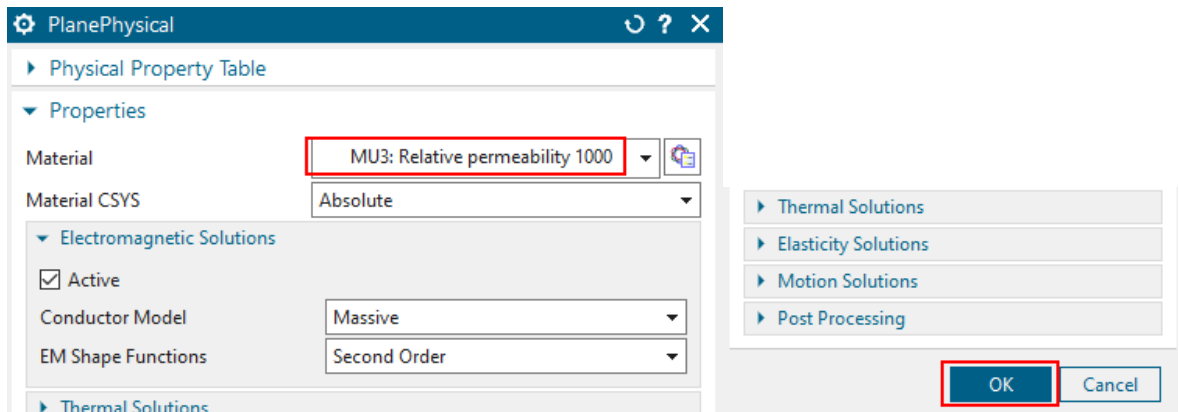


#### 5. Assign Properties to the Core

- Double click (or RMB Edit) the newly created 2D Collector. Set the Name to 'Core' and click 'Edit' . The window 'PlanePhysical' appears.




- In the 'PlanePhysical' window (below), select from the material list 'MU3...', a magnetic material with relative magnetic permeability value of 1000.
- accept the default Conductor Model 'Massive'.
- Hint: You can use the 'Manage Materials' feature to verify the 'relative Permeability (mur)' of this material is correctly set to 1000.

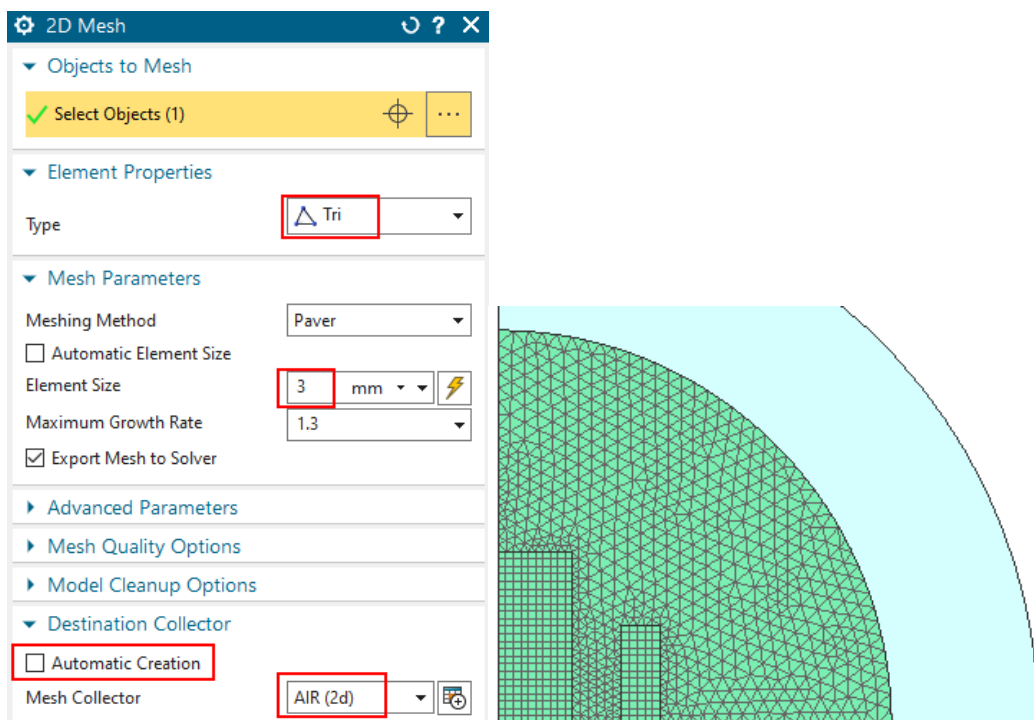


- Click OK, OK, to finish the definition of the core.

## 6. Mesh the inner air

Hint: In most cases air meshes are created as the last meshing step.


- Click on '2D Mesh'  and select the inner air face.
- Set the element type to 'Tri' and the 'Element Size' to 3 mm.
- At 'Destination Collector', clear 'Automatic Creation' and verify that the 'Mesh Collector' is set to 'AIR (2d)'.  
Hint: This collector is ready to use and comes from the Fem template.
- Click OK and the mesh is created using the correct collector and material.

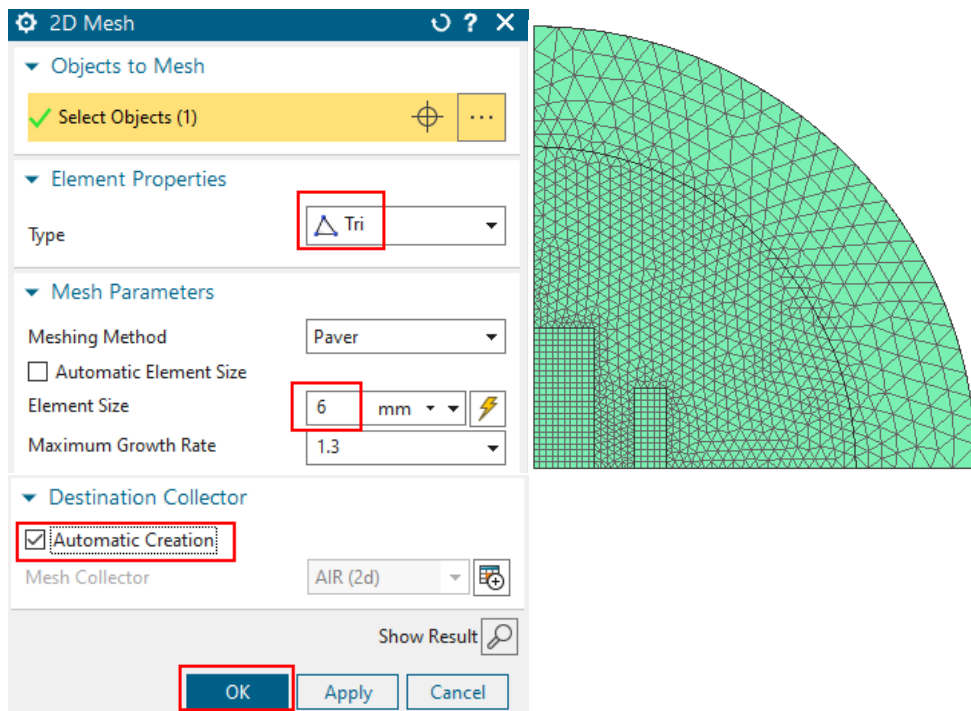


## 7. Meshing the Infinity Air:

- Background: The air around the core and the inductor geometry must be meshed to realistically model the magnetic (and in other cases the electric) field around these

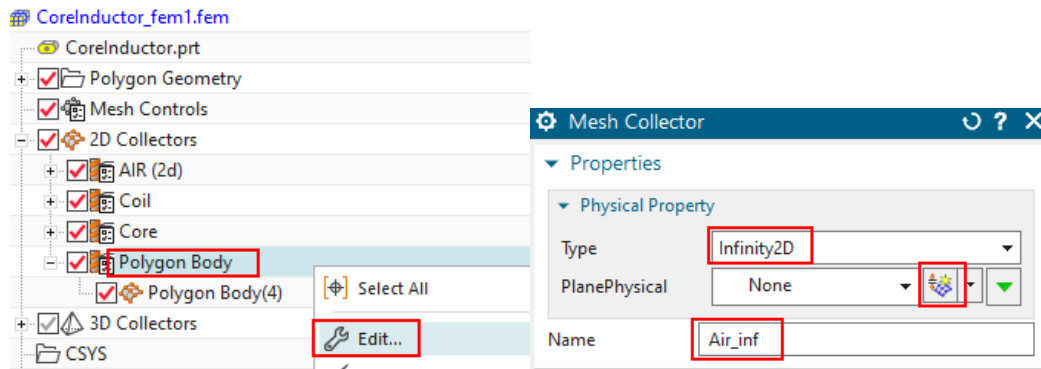
parts. This air geometry must be large enough to model the field. If it is too small, the simulation will model this field too "compressed" and the results may be inaccurate. To decide on the magnitude of this air geometry, one should also consider whether the field "wants" to leave the core and inductor geometry (flux leakage): if there is a closed iron path holding the magnetic flux, we would expect a small leakage and a large ball would not be necessary there. In our example, there is no such closed iron path and therefore we expect more leakage flux and the air geometry must be larger. Instead of a big-air geometry, there is also the option of using infinity elements. These infinity elements are to be used in our example. We used circular geometry, but other shapes like blocks would work as well. The advantage of a circular geometry is that this form is, so to speak, natural for the field lines. This also makes it easy to use Infinity elements.

- Click on '2D Mesh'  and select the outer air face.
- Use Tri elements of 6 mm size.
- At 'Destination Collector', activate again 'Automatic Creation'.
- Click OK and the mesh is created.

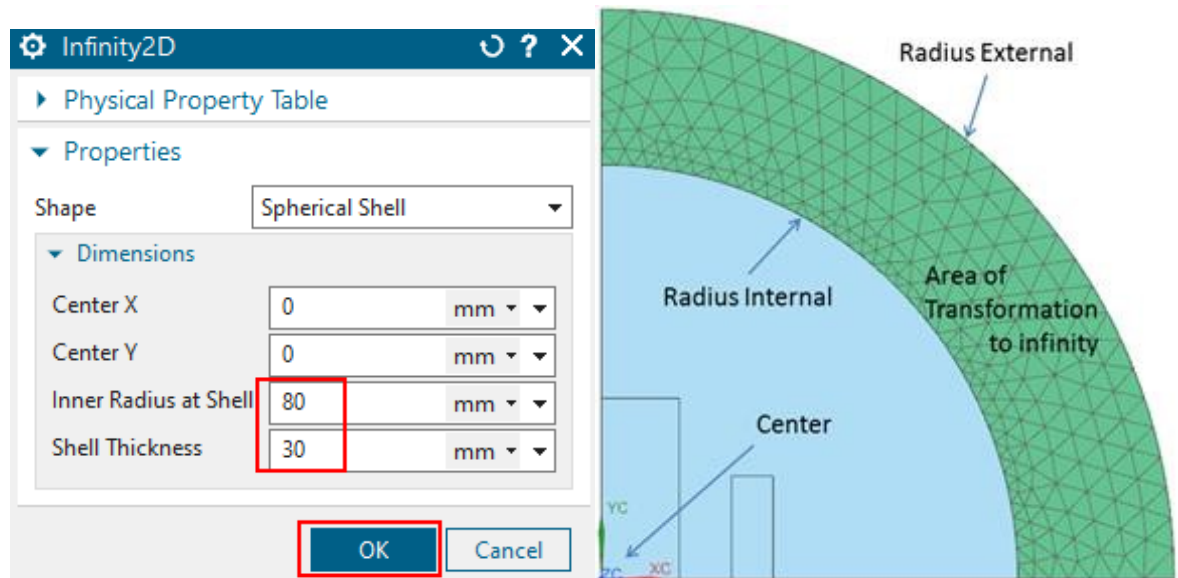


#### 8. Assign Properties to the Infinity Air


- Double click on (or RMB Edit) the newly created mesh collector.
- Set the Type to 'Infinity2D' and the Name to 'Air\_Inf'
- Click on 'Create Physical'. A new window appears.

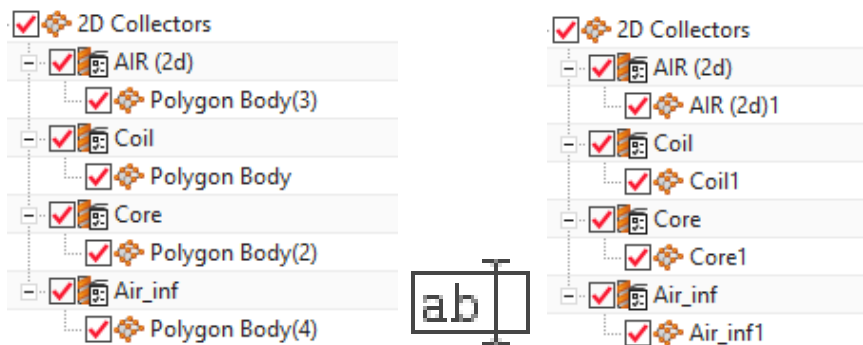


- in the window 'Infinity2D', key in the value 80 mm for 'Inner Radius at Shell' and 30 mm for 'Shell Thickness'. as shown in the picture. The dimensions must correspond to the size of the infinity air layer (see picture below).



Click OK, OK to create the Physical.

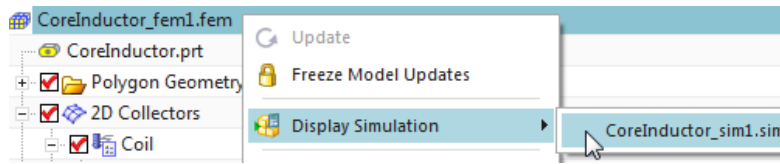
- As the final step in the Fem file, click on the button 'Auto Rename'  from the Magnetics toolbar. That utility renames all meshes and Physicals depending on the collector-names. It is a useful feature, especially with larger models and many meshes. Post processing becomes easier, thus we recommend this always to run after the meshing is done.




The picture above left side shows prior, right after the auto-rename.

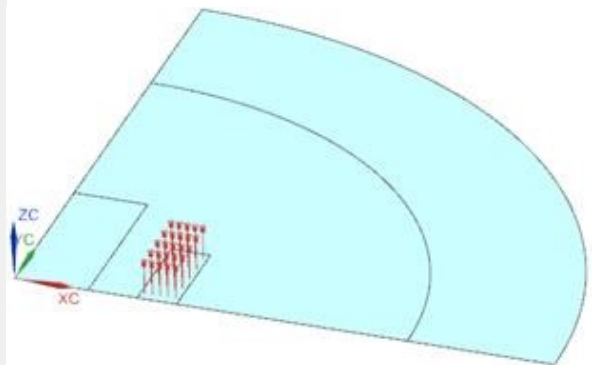
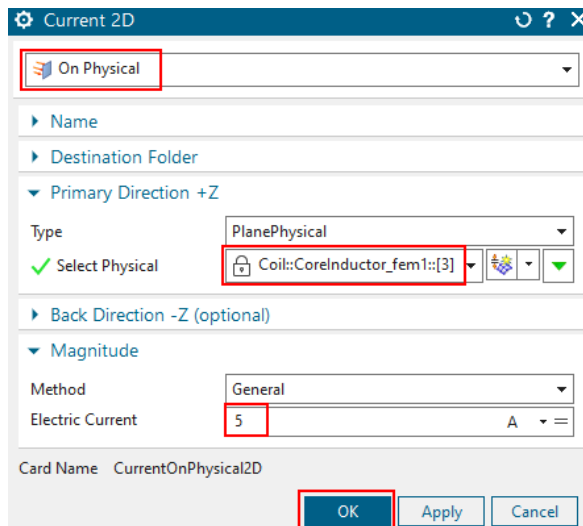
#### 4.1.4 Sim File Steps

1. We are now going to create boundary conditions. Therefore, set the Sim file to the displayed part. (The meshes are blanked here for easier visibility)



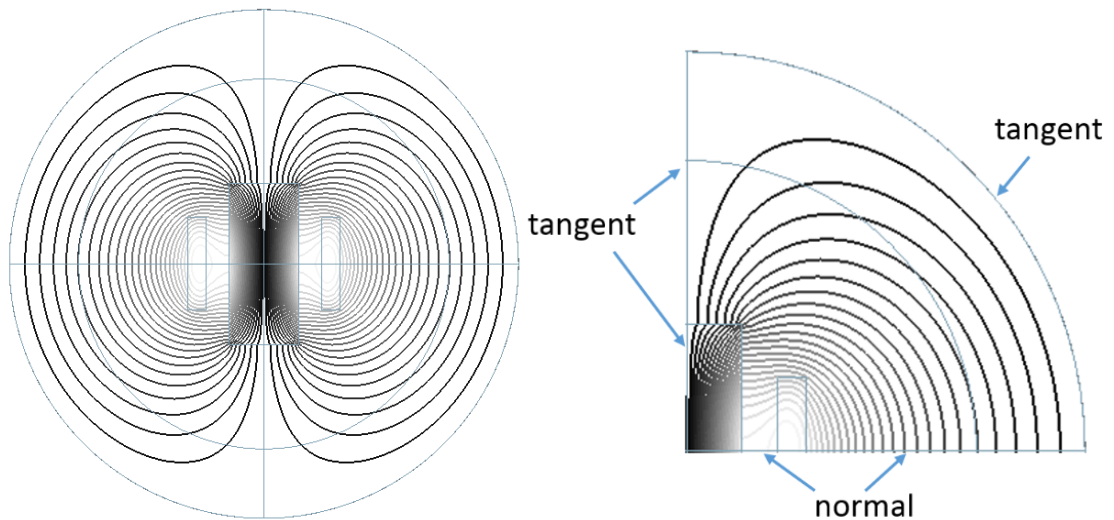
2. Create a load for the electric current on the coil:

- Click on 'Current 2D'  from toolbar Home→Loads and Conditions→Load Type.
- Accept the default Type 'On Physical'.
- At 'Primary Direction +', 'Select Physical', select the Coil Physical from the list.
- Assign the value 5 A for 'Electric Current'.
- Click OK and the load is created.




3. Understanding constraints

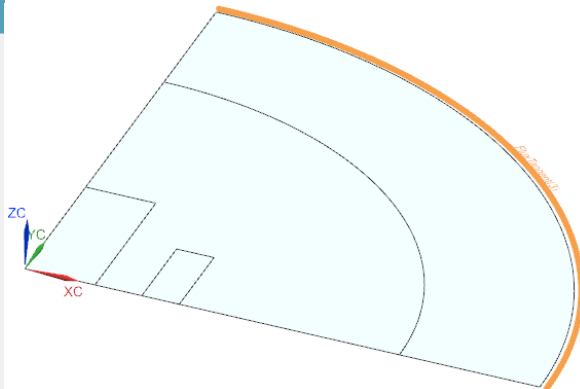
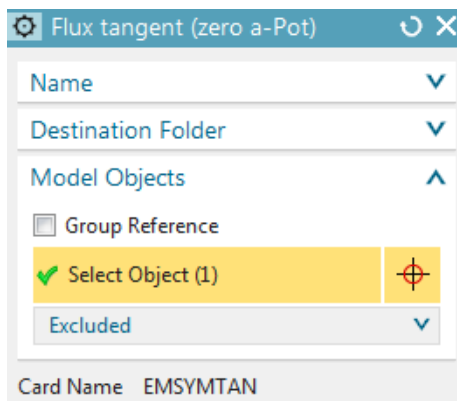
- The model we use represents a quarter of the full model. Therefore, we must correctly apply constraints at all borders. The picture below left shows how magnetic field lines would appear in the case of a full model.
- This allows understanding how constraints must be set in our quarter model: We need tangent magnetic field conditions at the circular border and at the vertical symmetry edges. And we need normal field conditions at the horizontal symmetry edges.
- Such field lines can also be shown in the results of the simulation: In 'Output Requests' activate 'Magnetic Potential (a-Pot)' and display this result in the post processor using the iso-lines option.




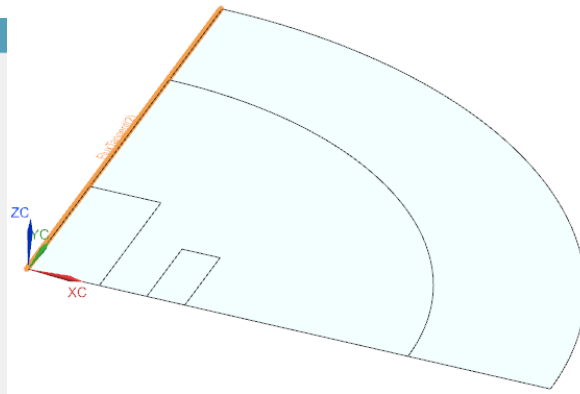
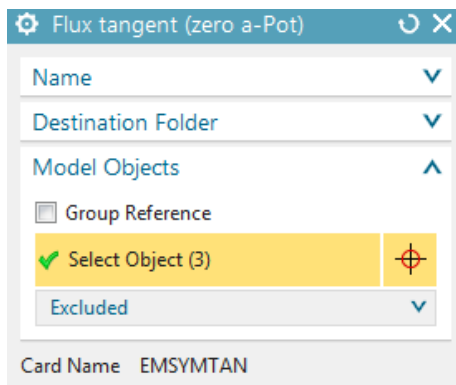
#### 4. Creating constraints.

According to such needs, now we will create two tangent and one normal constraint.

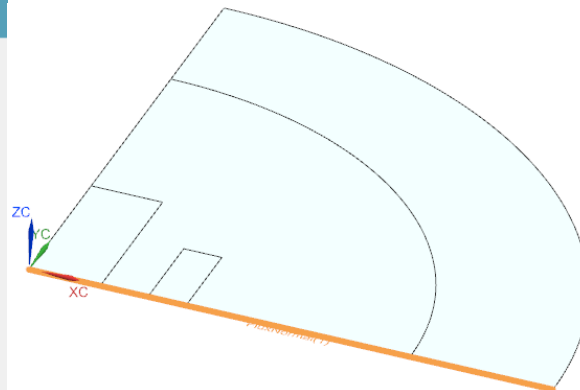
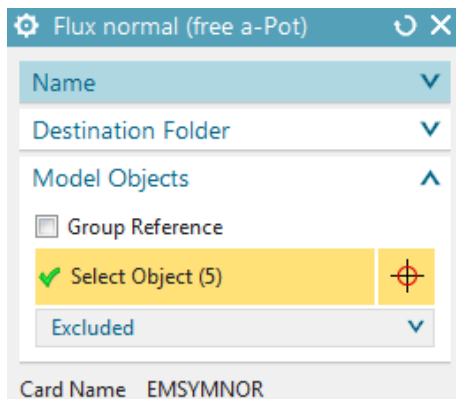
- Click on 'Flux Tangent (zero a-Pot)'  from 'Constraint Type'. Select the outside circular edge and click OK.
- Hint: This constraint type must always be applied at the outside air border to keep the field from leaving the computation area.




- Again click on 'Flux Tangent (zero a-Pot)'  and select the three  $x=0$  symmetry edges. Click OK.
- Hint: Viewing the resulting magnetic field lines helps understanding the symmetry condition.

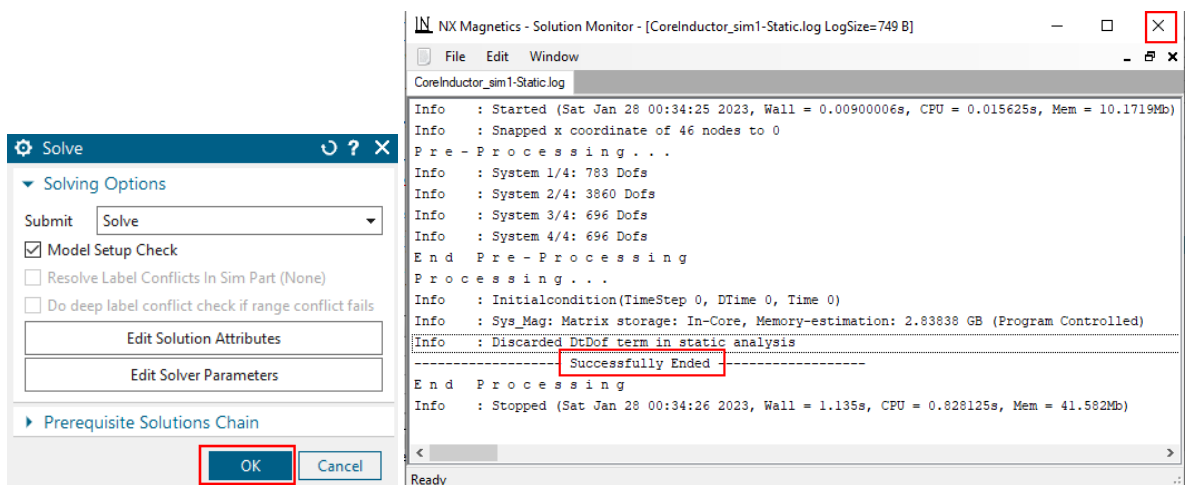


- Click on 'Flux normal (free a-Pot)' and select the five y=0 symmetry edges. Click OK to create the condition.



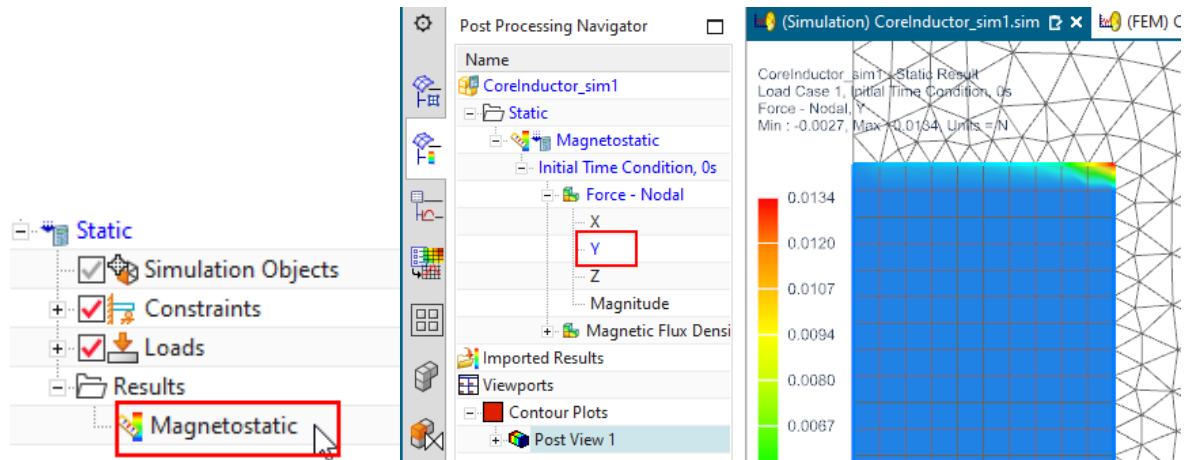
5. Solve the solution:


- Click on 'Solve'  from toolbar 'Solution'.
- click OK. The solve job starts and a new window appears to monitor the solution progress. The solve job takes only a view seconds for this small model.
- If the solution monitor shows 'Successfully Ended', the job has ended and there should be results. Cancel the solution monitor and also other info windows that may have appeared.

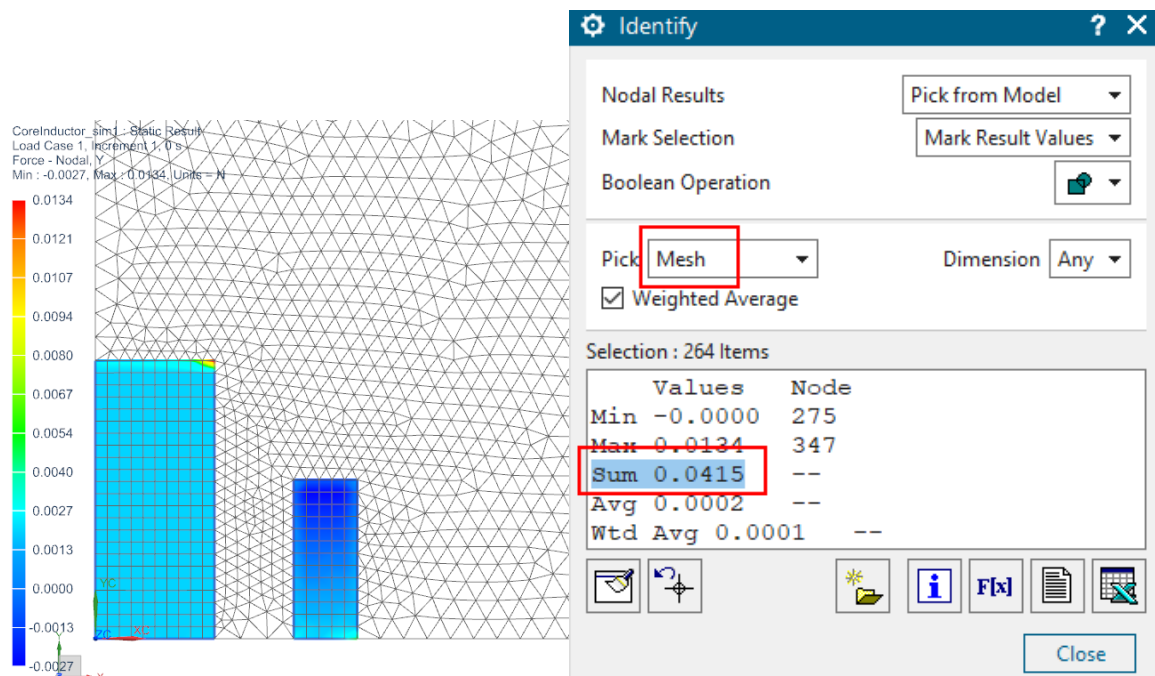


6. Post Processing Force Results

- Double click on the 'Result' node in the Simulation Navigator.  
Hints: The system opens the result file and switches to the post processor. The field results are managed using the 'PostProcessing Navigator' (see below right).



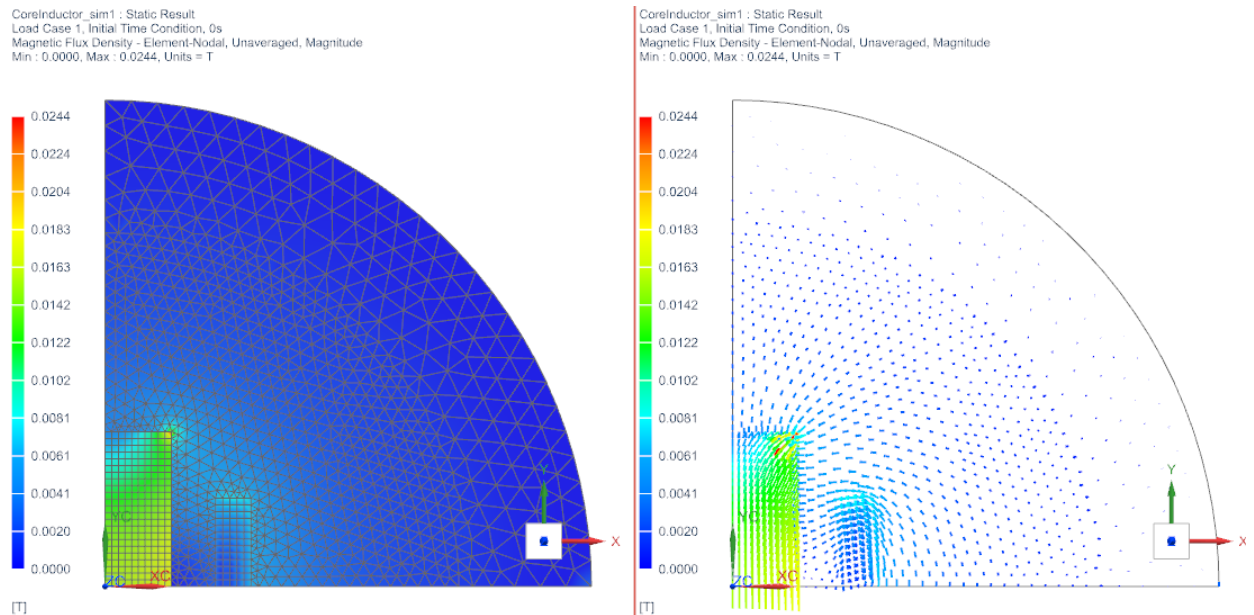
- Double click on 'Force - Nodal Y' to plot the force distribution in Y direction. There appears a PostView showing this.
- Click on 'Identify Results'  from toolbar 'Results'
- set 'Nodal Results' to 'Pick from Model' and option 'Pick' to 'Mesh' and select the face of the core.
- The sum of all values on the core is evaluated and written into the 'Identify' window. This resulting force sum is about 0.04 N.  
Hints: The defined thickness value of 150 mm is already taken into account.



## 7. Post Processing Magnetic Flux Density

- To plot the distribution of magnetic flux density double click on 'Fluxdensity - Element Nodal Magnitude'.

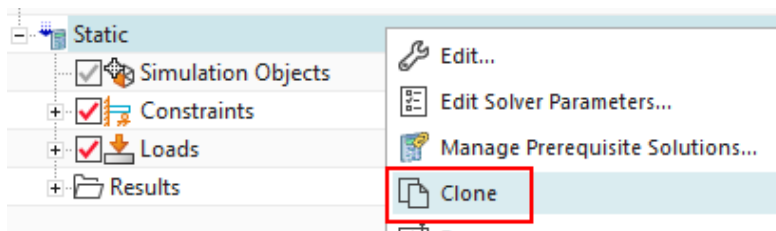
- to display the magnetic flux density by arrows, switch from 'Contour' to 'Arrows' in toolbar 'Results'. These vectors show the direction of the field lines.



#### 4.1.5 Axisymmetric Analysis of Core and Conductor

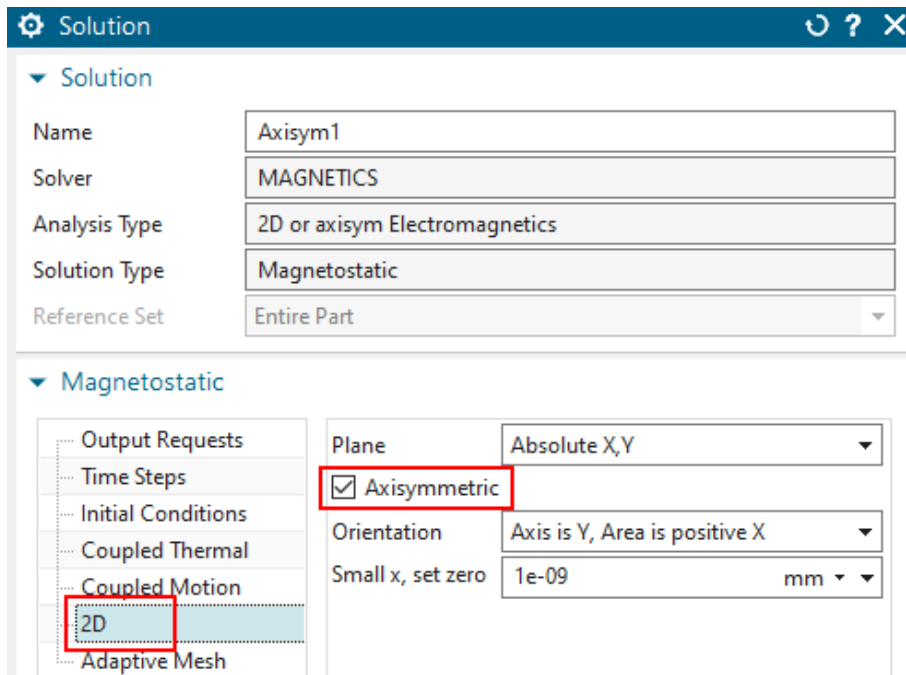
To analyze in an axial symmetric way (axis is Y) using the same geometry follow these steps:

1. Clone the solution and
2. Rename the new solution to 'Axisym1'.



3. Choose 'Edit Solution' on the new 'Axisym1' and switch to register '2D'. Activate 'Axisymmetric' and click OK.

Hint: To use axis symmetric analysis, it is necessary to place all elements on the x/y plane for positive x. The axis must be Y.

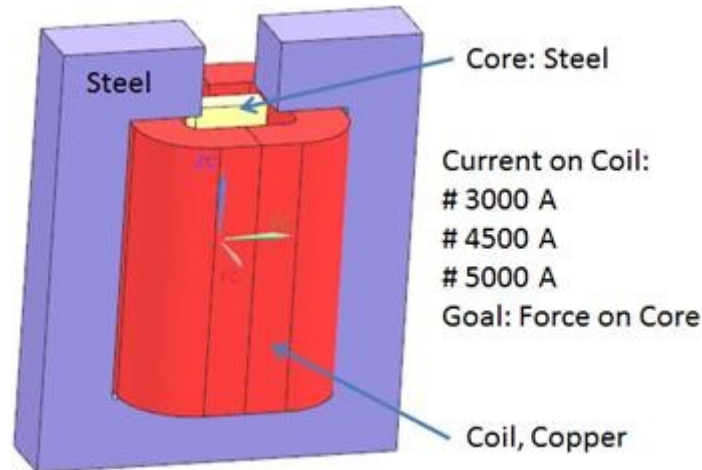


4. Solve the solution  $\begin{bmatrix} + \\ - \end{bmatrix} =$  and post process the results:
5. Check the force sum in Y direction again. It should be about 0.08 N. The axisymmetric effect ( $2 \cdot \pi$ ) is already taken into account.
6. The tutorial is complete. Save your files and close them.

## 4.2 Static Force on Solenoid (TEAM 20) (Beginner)

This example is a convenient training for all those who want to do electromagnetic analysis with complex 3D geometries, because it contains most necessary skills for dealing with this.





The Team20 benchmark is a test example for electromagnetic analysis software tools. There are measured force-results available that we will use for comparison with the simulation-results. We will use a nonlinear B-H curve that includes saturation effects for the core and iron parts.



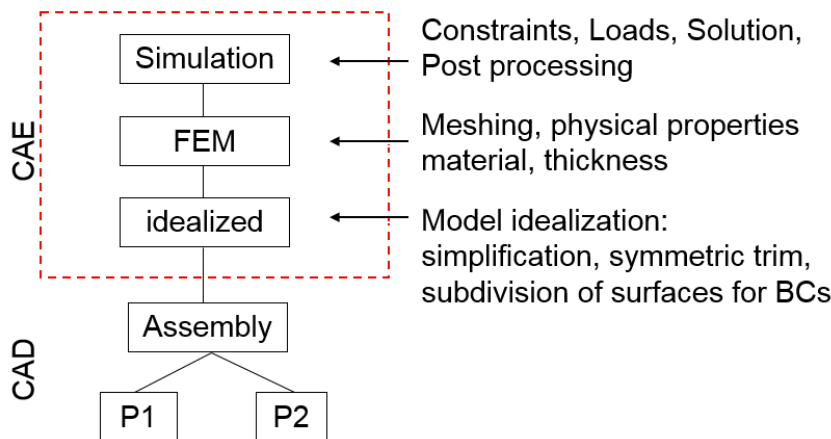
For meshing, we want to exploit the 'Non-Manifold' strategy, which automatically finds adjacent face pairs and creates conformal meshes (e.g. coincident nodes) at those interfaces. Thus, no Mesh-Mating conditions are needed any more. Unfortunately, this must be done differently in the different NX/Simcenter versions. Prior to NX 1953 (2021.1) this approach is not possible at all and must be replaced by the conventional Mesh-Mating method.

Necessary time: 1 h.

### 4.2.1 Preparations

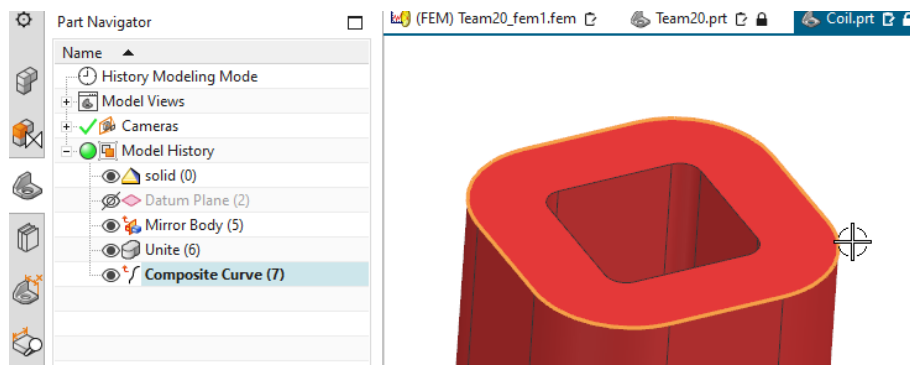
1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/3.MagSta/3.2Team20.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program NX  or Simcenter . Use Version 12 or higher.
4. In NX/Simcenter, click Open  and navigate to folder 'start'. Select the file 'Team20.prt' and click OK. (Maybe you must set the file filter to 'prt')
5. From toolbar 'Application', click on 'Pre/Post' .  
The system switches into the simulation application.
6. Hint 1: All information describing the simulation is managed in basically three files (see below): The **Sim**, the **Fem** and the **Idealized** file. The Sim file contains the solution, boundary conditions, loads. The Fem file contains the meshes, material properties and other physical properties. The Idealized file is optionally and can be used for simplifications

or modifications of the geometry without changing the original CAD file.

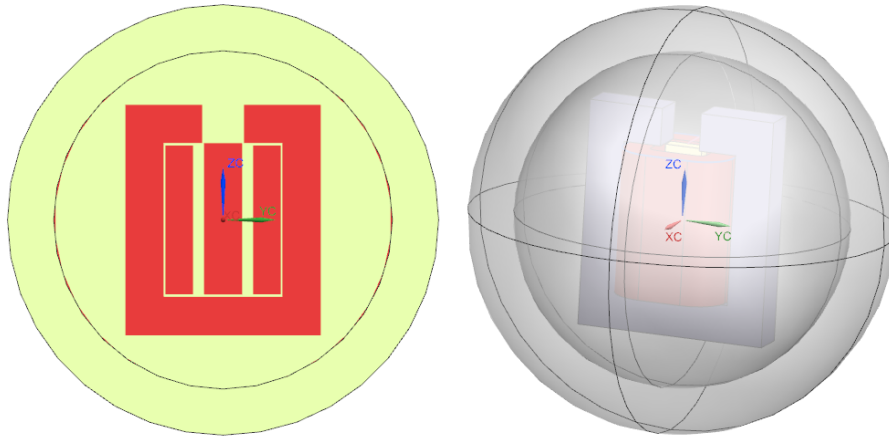


Following, we will first create this file structure and then we will fill it up. The 'Simulation Navigator' will allow us navigating through the structure.

7. Hint 2: Notice, in the CAD Model, there is a **spline curve** extracted from the outside edges of the coil (see highlighted in below picture). This spline will be used to define the winding directions. It is necessary, that this spline is one single curve. In case there are multiple curve segments, there must be done a 'Join Curve' operation to make one spline out of them. We will later have to activate one special button to transfer this spline into the Fem file.



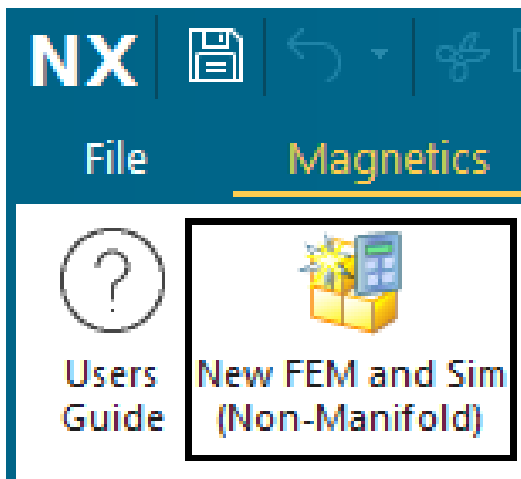
8. Hint 3: A **spherical air volume** is placed around the solid geometry and an additional spherical shell is placed around the sphere. This sphere is necessary to model the magnetic (and in other cases also the electric) field in air. The shell additionally models the infinity air. We could solve without the infinity air, but in that case the air volume would have to be large enough to model the field. In case it would be too small, the simulation would model that field too much 'compressed' and results would become inaccurate. To decide about the size of such an air volume, it should also be considered whether the field 'wants' to leave the solid geometry (flux leakage): If there is a closed iron path, holding the magnetic flux like in our case, there is no need for a big sphere. In other cases, there may appear more flux leakage and the air volume should be larger. Instead of using a large air volume, we rather want to use infinity elements just like we already did in the previous tutorial 'Core and Inductor'. In this example, we use a sphere, but alternatively box geometries would work as well. Advantage of a sphere is that it is kind of natural for the field lines. Also this makes it simple using infinity elements.



#### 4.2.2 Create Fem and Sim Files

This step must be done differently in the different NX/Simcenter versions because of the non-manifold feature we want to use. For more details, see chapter 'Recommended System Settings'.

1. **Version 2212:** In toolbar Magnetics, click 'New Fem and Sim (Non Manifold)'. This runs a script to create the Fem and Sim files from templates with non-manifold feature.




2. **Version 2007 (2022.1) and 2206 (2022.2):**

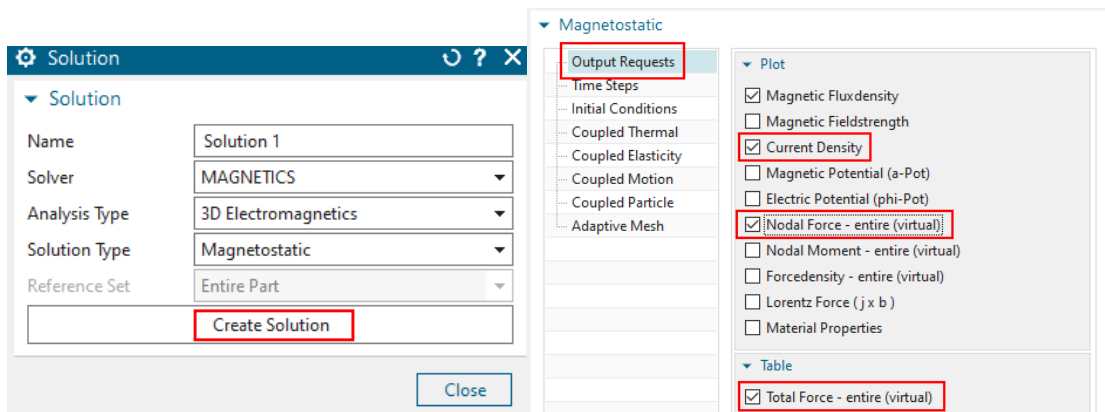
- Activate the customer default 'Create Non-manifold Polygon Bodies'.
- Create a 'New FEM', select the MAGNETICS template.
- Create a 'New Simulation', also select the template.
- The creation of the solution continues below.

3. **Version 1953:**

- Activate the early access feature 'Allow updates related to Electro Magnetics'.
- Create a 'New FEM', select the MAGNETICS template.
- Create a 'New Simulation', also select the template.
- The creation of the solution continues below.

### 4.2.3 Create the Solution

1. In toolbar 'Home', click on 'Create Solution' 
2. Accept the Solver 'MAGNETICS' and the Analysis Type '3D Electromagnetics',
3. Accept the default Solution Type 'Magnetostatic'.
4. Click 'Create Solution'.
5. In register 'Output Requests', 'Table', activate 'Total Force - entire (virtual)' to enable the system calculate forces by the virtual energy method.
6. to see these forces in the plots and also the applied currents, also activate in 'Plot' 'Current Density' and 'Nodal Forces - entire (virtual)'.

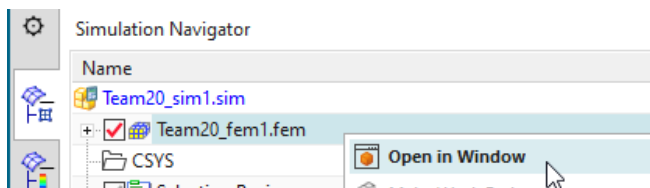



7. Click OK to finish the Solution window.

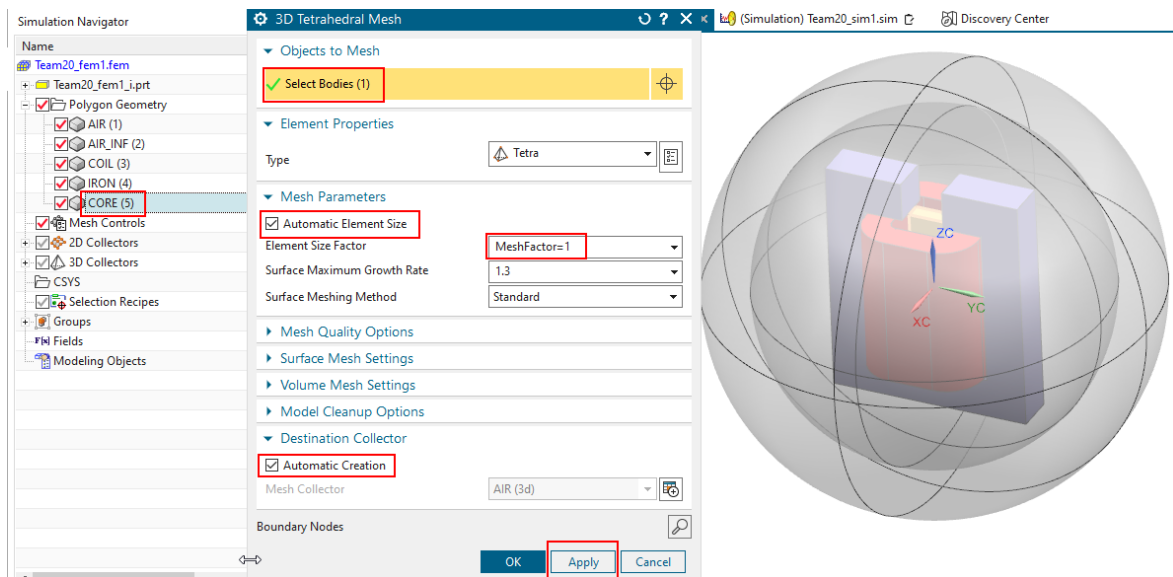
### 4.2.4 Parametric Meshing

Following, we will create all element-sizes using one control-parameter 'MeshFactor'. This makes it simple later to do studies with different mesh-sizes and therefore, we recommend this approach.

1. Do a 'Open in Window' to the Fem part.



2. Mesh the CORE: Start the mesher '3D Tetrahedral' , select the core body (either in the graphics window or in the Polygon Geometry list) and activate 'Automatic Element Size'. In the field 'Element Size', key in 'MeshFactor=1' to create and initialize the parametric expression. Check, that 'Destination Collector' is 'Automatic Creation' and click 'Apply'. The mesh is created.

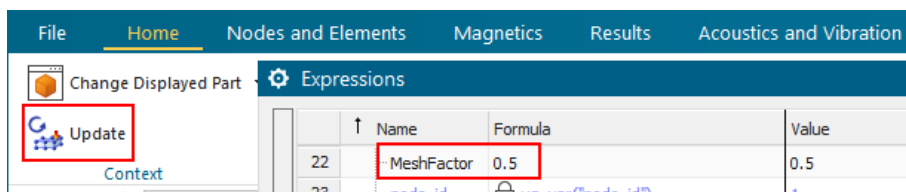


3. Mesh the remaining parts with nearly the same settings as follows:

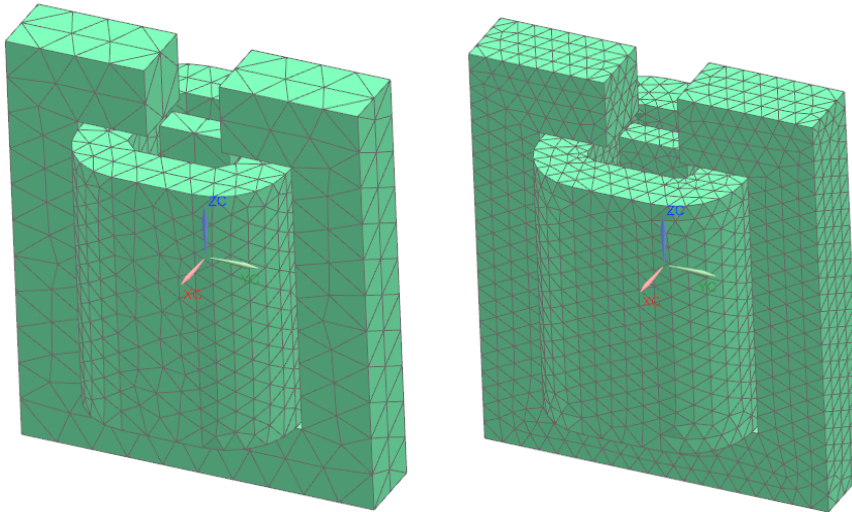
- Select the IRON body, click 'Apply'.
- Select the COIL body, click 'Apply'.
- Select the AIR body, set 'Mesh Collector' to 'AIR (3d)', click 'Apply'.
- Select the AIR\_INF body, click 'Apply'.

Hint: the order of creation of meshes is quite important: Following meshes must always connect to the existing nodes of prior meshes. Thus, we should always start with the most important parts.

4. The meshes are still quite coarse, because of the MeshFactor=1. We want to globally reduce the mesh size to the half. Thus, change the expression 'MeshFactor' to 0.5. This is done in the 'Expressions' window (Menu, Tools, Utilities, Expressions).





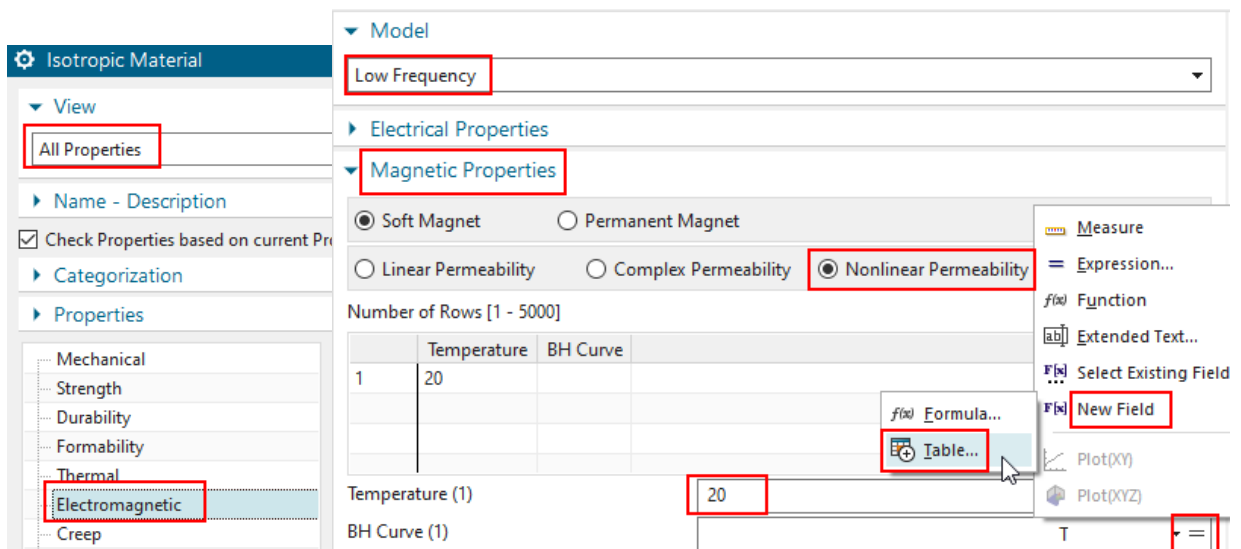
After the new value is set, click the 'Update' button and all elements will become smaller. The picture below shows left the old and right the new meshes of core, iron and coil.



#### 4.2.5 Define Core/Iron Material with Nonlinear B-H Curve

In this chapter, we define a typical steel material that is used for the core and iron bodies. This material shows a nonlinear relationship between magnetic flux-density ( $B$ ) and magnetic field-strength ( $H$ ). Such a relation is often the case for steel and iron. Many material properties are already available in the material library, but often, we must insert individual data and define a individual material. The data points with H-B pairs are stored in a csv file (Team20MaterialCurve.csv) in the 'start' folder of this tutorial. Proceed as follows to create a new material with these:

- Click 'Manage Materials'  and 'Create Material'  at the bottom right.
- Set 'View' to 'All Properties' and select the register 'Electromagnetics'. Key in a name, for instance 'Team20 Material'. Set the 'Model' to 'Low Frequency' (see below).



The screenshot shows the 'Isotropic Material' dialog box. The 'View' dropdown is set to 'All Properties'. The 'Model' dropdown is set to 'Low Frequency'. The 'Magnetic Properties' section is expanded, and 'Nonlinear Permeability' is selected. A table with 3 columns (Temperature, BH Curve) and 1 row (1, 20) is shown. The 'Temperature (1)' field is set to 20. The 'BH Curve (1)' field is set to T. The 'Table...' button is highlighted.

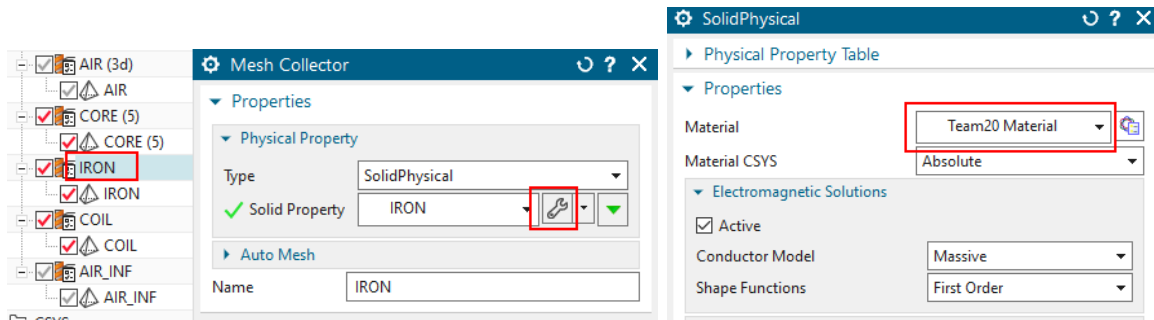
	Temperature	BH Curve
1	20	

- Scroll down and set 'Magnetic Properties' to 'Nonlinear Permeability' (above right). Key in a 'Temperature' of 20 degrees Celsius.

- Right to field 'BH Curve', select the '=' and choose 'New Field' and click 'Table'.
- In the next window, select 'Import from Text File' and scroll to the 'start' folder of this tutorial. Select the file 'Team20MaterialCurve.csv' and click OK. The picture below right shows the 'Table Field' window with the new data.

Row ID	magnetic field strength (A/m)	magnetic flux density (T)
1	0	0
2	27	0.01
3	58	0.025
4	100	0.05
5	153	0.1
6	185	0.15
7	205	0.2
8	233	0.3
9	255	0.4
10	285	0.5

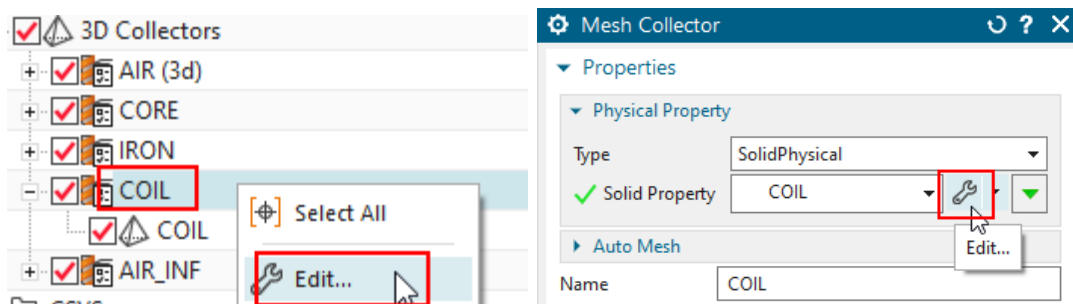
- Click OK, Close. The material is now created.
- Assign the new material to the core and also to the iron:



#### 4.2.6 Define Stranded Coil Properties

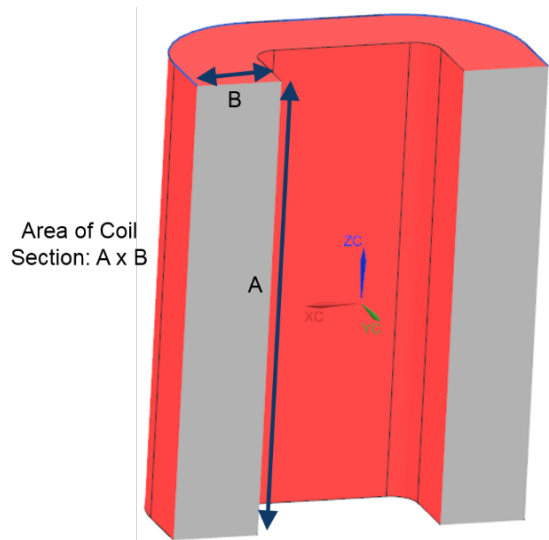
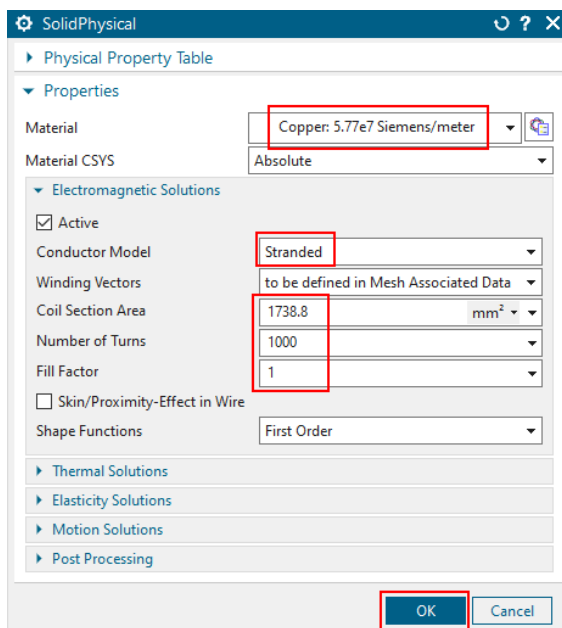
Following, we will define the properties of the coil. Even though the CAD model is a simple solid, in the simulation we want to define the electric current as flowing through individual turns.

1. Edit the Physical of the coil mesh.

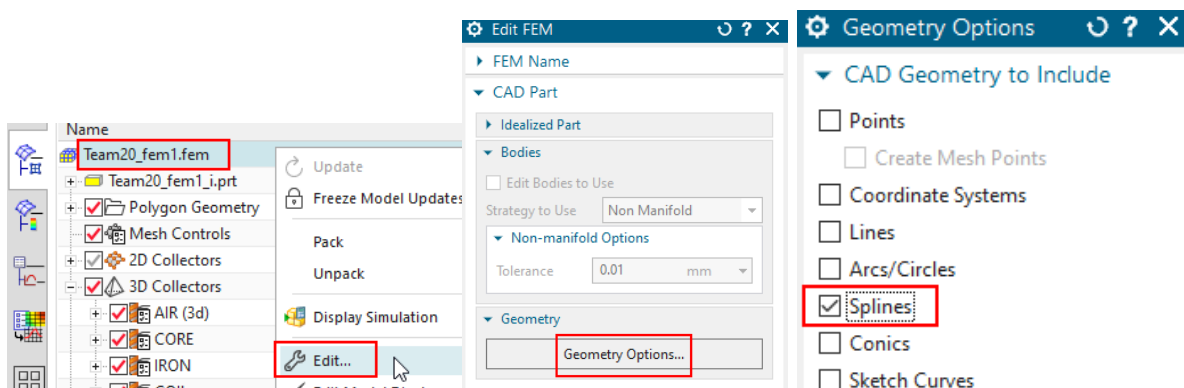


2. In the box 'Conductor Model', set the options as follows:
3. Set 'Model' to 'Stranded, Vectors defined in Mesh Associated Data'. This allows forcing all electric currents flowing in directions of the winding. Using this stranded method you will get a homogeneous distribution of the current over the face. In a following step we will define these directions using a spline curve.

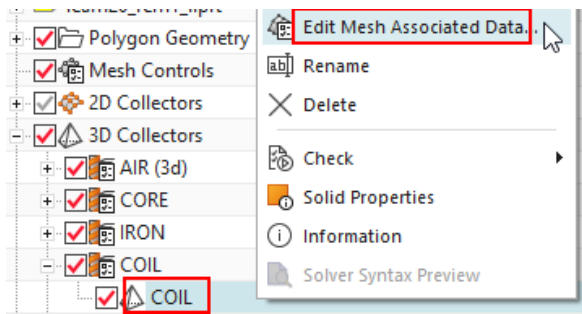
4. At 'Material', choose the copper from the list. (There are already some basic materials in the list because of the template that was used at creation)
5. Set 'Number of Turns' to 1000,
6. Accept 'Fillfactor' is 1, This setting influences the ohm resistance of the coil.
7. Set 'Coil Section Area' to 1738.8 mm<sup>2</sup>. See next picture (right) for an explanation how this area is computed.



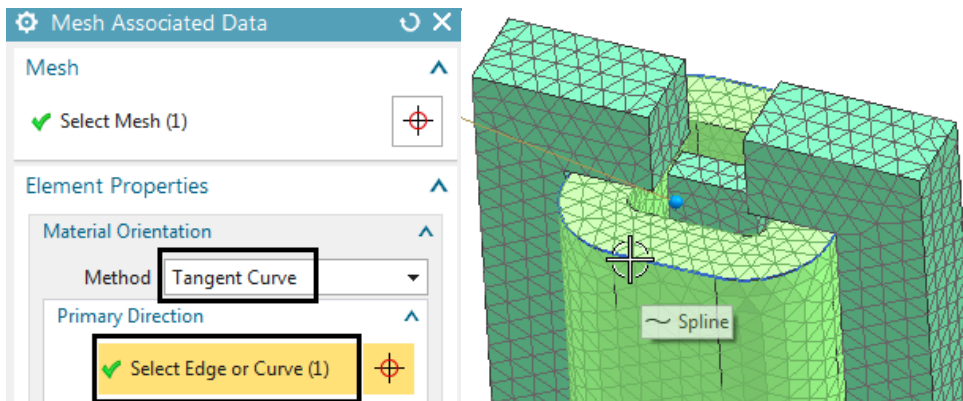
8. Define the winding directions. We first have to add the spline curve into the Fem file, which in CAD was extracted from the coil edges. To do this: edit the Fem file, click 'Geometry Options' and activate 'Splines' in the following window. Click OK, OK.



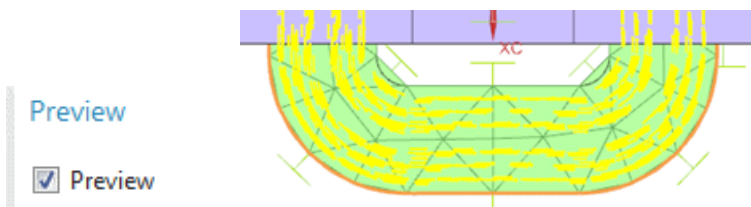
9. Now, do RMB on the mesh of the coil, Choose 'Edit Mesh Associated Data'.



10. In the next window, set the option 'Material Orientation' to 'Tangent Curve' and select the spline curve for the stranded winding direction. (Blank the meshes and air volumes for easier selection)




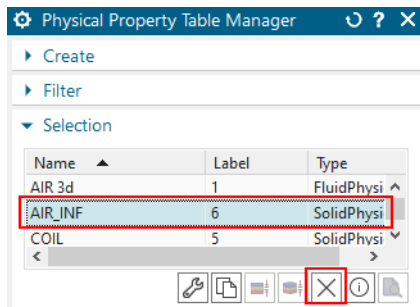
11. Hint: Maybe, the spline curve is not visible. In that case, you have to fully load the CAD file of the coil. Therefore, simply make the coil the work part once.
12. Hint: You can activate 'Preview' to see the arrows pointing in positive direction. If later, current is defined on this coil, it will be forced in this direction.



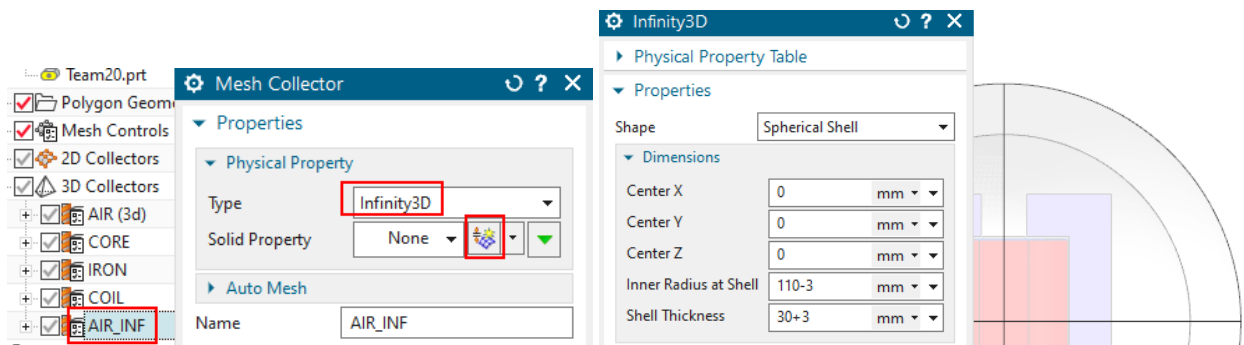
13. Click OK, OK to finish the coil definition.

#### 4.2.7 Define Infinity Properties

1. First, delete the existing Physical named 'AIR\_INF', because it has been automatically created with type 'SolidPhysical', but we need one with type 'Infinity3D': From toolbar 'Home', 'Properties', choose 'Physical Properties' , select the 'AIR\_INF', which is of SolidPhysical type and click 'Delete' and 'Close'.



- Now, create a correct Physical: Edit the mesh collector 'AIR\_INF', set the 'Type' to 'Infinity3D', select 'Open Manager', accept the default type 'Spherical Shell' and key in the two values for 'Inner Radius at Shell' and 'Shell Thickness' as shown below. Click OK, OK.

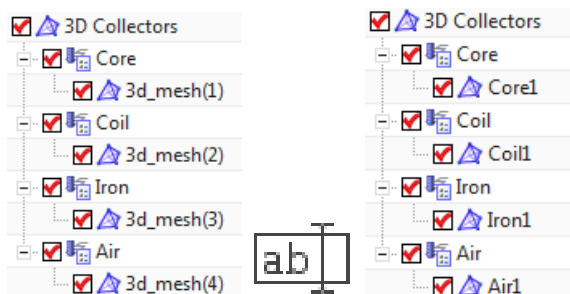


Hint: The value 'Inner Radius at Shell' is slightly reduced (3 mm, and the thickness value adds 3 mm to compensate) to avoid error messages like 'Bad parameters for transformation Jacobian' that may appear from the solver. The reason for that error is that all elements of AIR\_INF must be inside the two radii. But if the mesh is coarse, there may some element-edges crossing the radius value. To allow also these elements, we do that reduction.

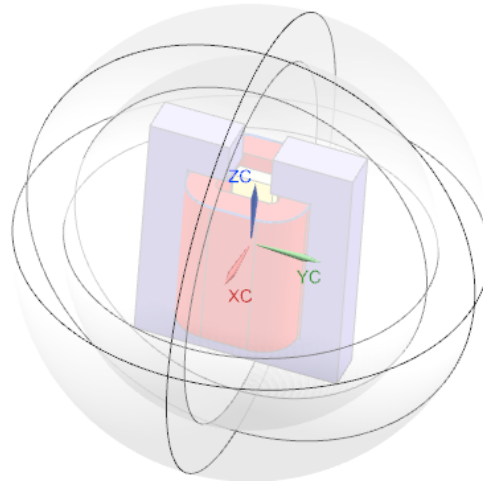
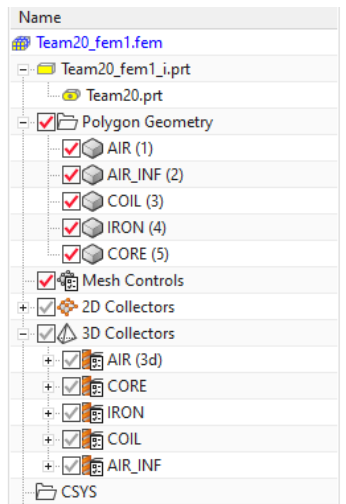
#### 4.2.8 Finish Fem File

This step is not necessary if in customer defaults, the setting 'Use Polygon Body Names for Meshes and Physicals' is activated, see chapter 'Recommended System Settings'.

- For easier post processing, click on the button 'Rename Meshes by Collectors' from the Magnetics toolbar. That utility renames all meshes and therefore is a useful feature especially with larger models and many meshes.

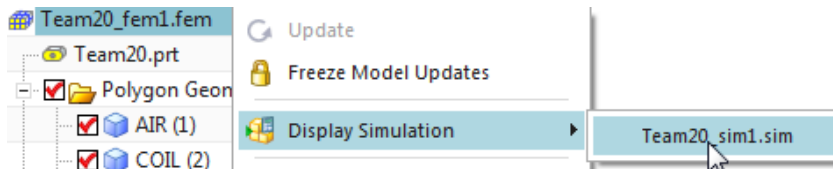


- To finish the work in the Fem file it is a good practice to blank all meshes and unblank all polygon bodies.



#### 4.2.9 Define Load and Constraint

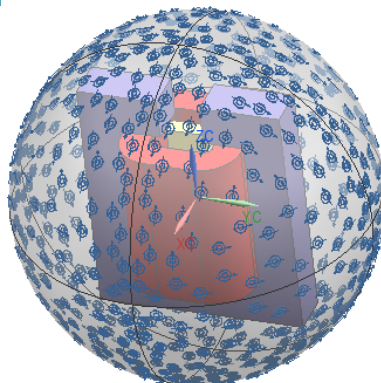
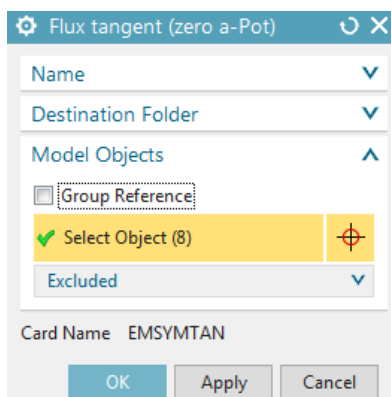
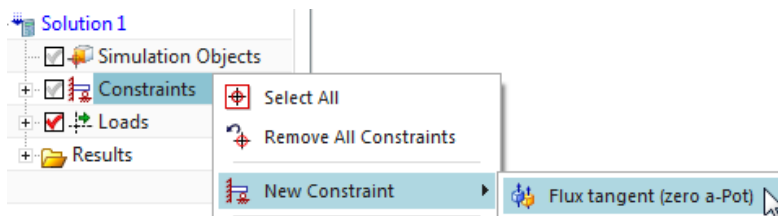
1. Switch to the Sim file.




2. To keep the magnetic flux inside the air sphere, it is necessary to use a constraint 'Flux tangent'. To do so, click on 'Constraints→'New Constraint'→'Flux tangent (zero a-Pot)'



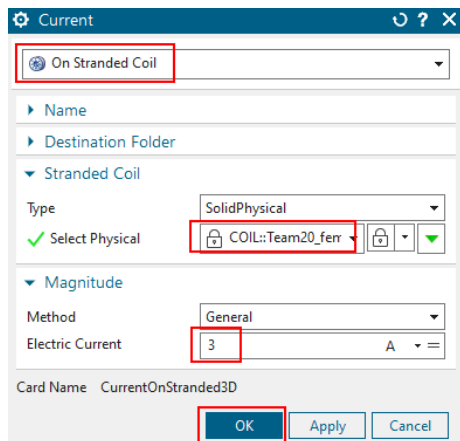
. Select all 8 outside faces of the air volume. Click OK.




3. To create an electric current load for the coil:

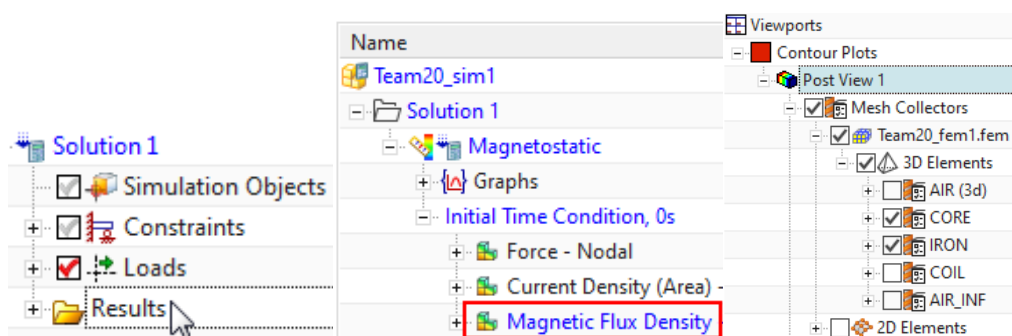
- At loads, click on 'Current' ,

- accept the default type 'On Stranded Coil' and also the default Method 'General'
- key in 3 A at 'Electric Current'.  
Hint: The positive direction of the current is defined by the spline that was selected in 'Mesh Associated Data'.
- In the list 'Select Physical', select the 'Coil' and click OK to finish the creation of the coil current.

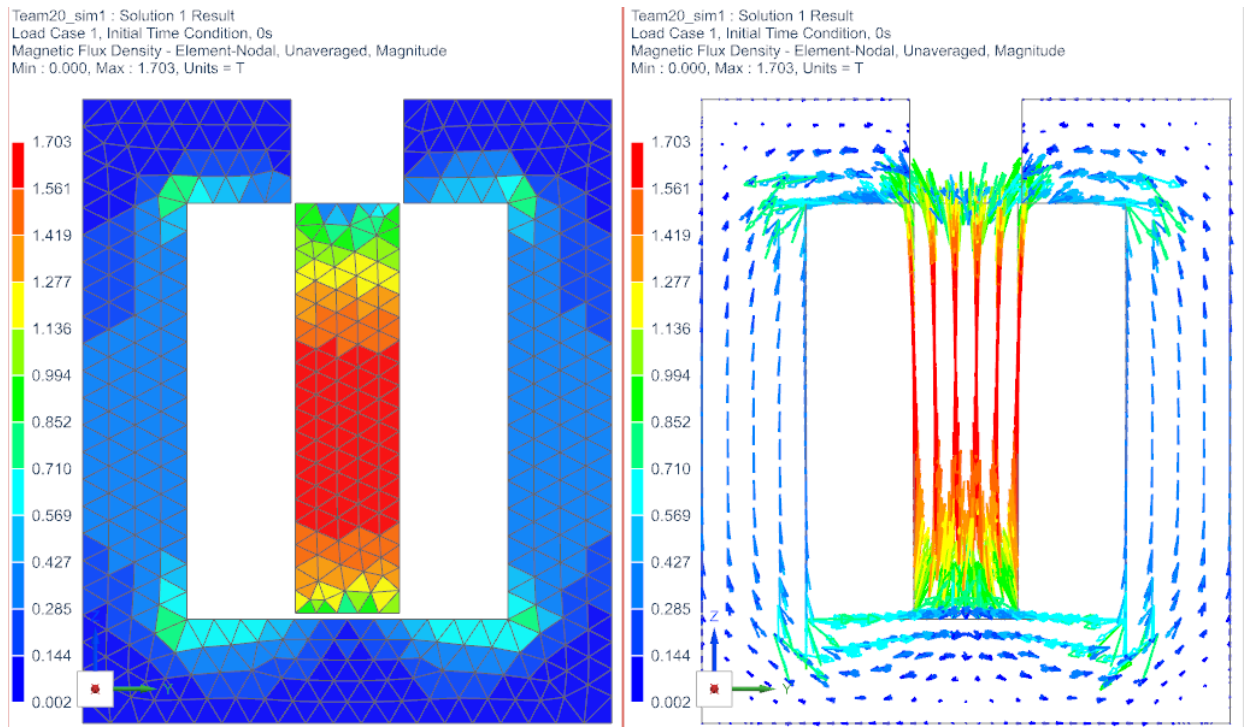


#### 4.2.10 Solve and Post Processing

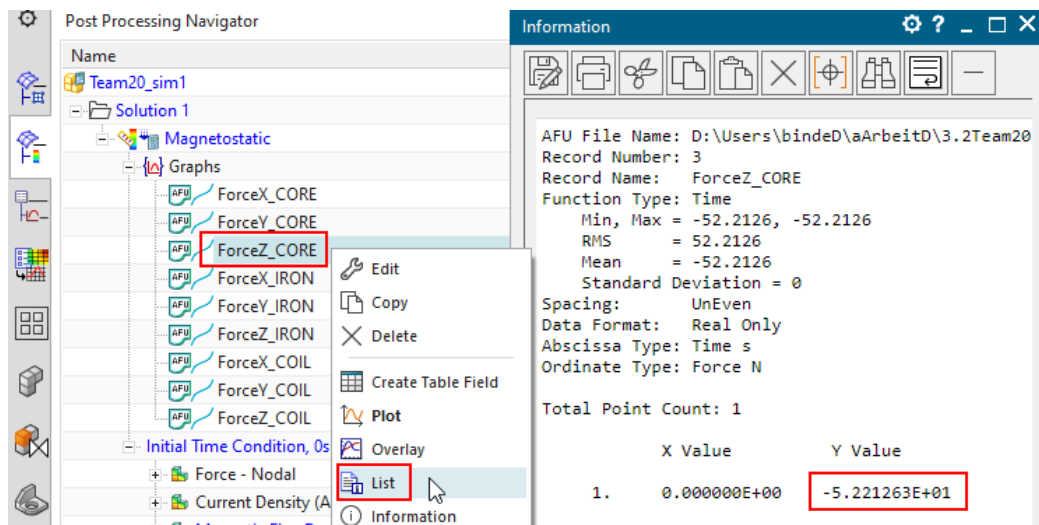
1. Solve the solution: Click on  and OK. The solver will run about 20 sec.
2. Post processing
  - After the solver has finished open the results by double clicking on the 'Results' node. Then, in the 'Post Processing Navigator', open the node 'Solution1' and double click on 'Magnetic Flux Density ...'.
  - Also in 'Post Processing Navigator', hide the AIR, AIR\_INF and COIL meshes by switching off their red symbols at 'Viewports'. Now the results can be seen in graphics.



- The maximum Flux density in the core should be about 1.7 Tesla. The below picture shows Contour (left) and Arrows (right).



- Tabular results, like force sum on all bodies or currents, are found at the top of the results node, as seen in the below picture. From there, the requested results can be opened and shown by 'List' or in a graph window by 'Plot'.



The Z-direction force is computed to -52.2 N.

Notice: The measured reference force is -54.4 N.

### 3. Variants with 3A, 4.5A and 5A

We want to compute two other electric currents and compare them with reference measurements. Solve using the following values for the current and compare the results against measurement data (Reference data is from [TakahashiNakataMorishige]).

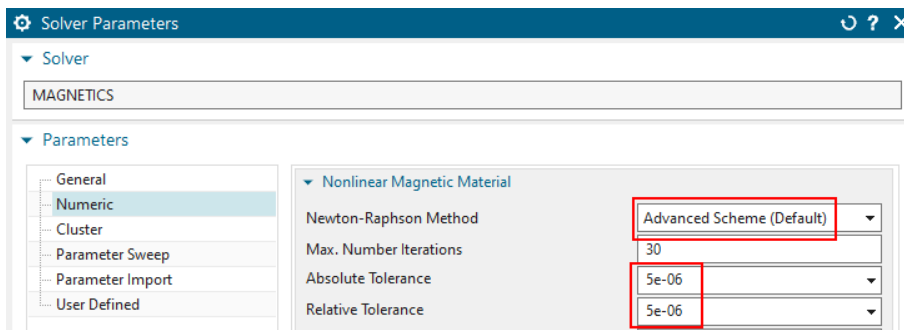
- 3.0 A: Measurement: -54.4 N. Simulation: -52.2 N Deviation: 4%
- 4.5 A: Measurement: -75.0 N. Simulation: -80.4 N Deviation: 7.2%
- 5.0 A: Measurement: -80.1 N. Simulation: -88.5 N Deviation: 10.4%

**Loss of Accuracy** Loss of accuracy normally results mainly from the mesh because of the approximation of finite elements. Thus, finer meshes or using higher order shape functions will help. On the other side, if there is a nonlinear material included, there also results some loss of accuracy from the nonlinear scheme what can be controlled by the residual tolerances. We will use both in the following.

#### 4.2.11 Increasing Accuracy by Reducing Nonlinear Residuals

Because we use a nonlinear BH curve in one of the materials, there runs a Newton-Raphson-Iteration-loop in the solver. Tolerances control how tight our solution will be to the given BH curve. Thus, the first thing we do is reduce this tolerance by one order. Proceed as follows:

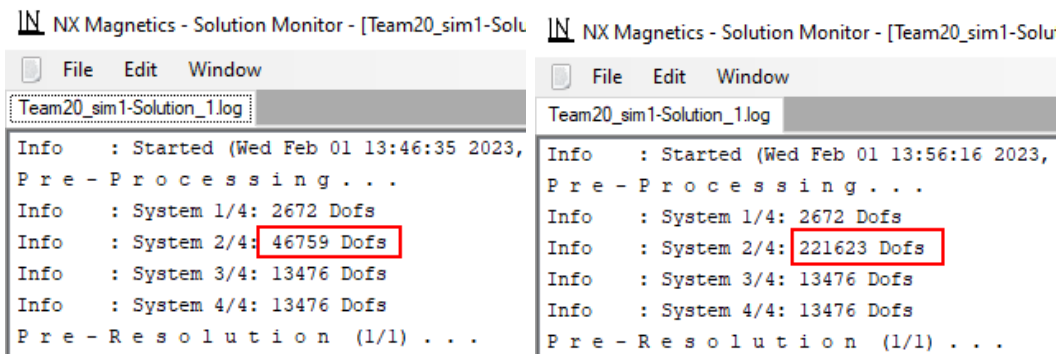
1. RMB on the solution, click 'Edit Solver Parameters'. In register 'Numeric', box 'Nonlinear Magnetic Material', set 'Newton-Raphson-Method' to 'Advanced Scheme (Default)'.
2. reduce the two tolerances by one order, thus, from  $5e-5$  to  $5e-6$ .



3. Solve again for the three currents and compare the force results. You may also notice in the logfile that the number of required iterations to converge increases.
  - 3.0 A: Measurement: -54.4 N. Simulation: -51.6 N Deviation: 5.1%
  - 4.5 A: Measurement: -75.0 N. Simulation: -75.1 N Deviation: 0.1%
  - 5.0 A: Measurement: -80.1 N. Simulation: -80.9 N Deviation: 1.0%
4. now we reduce the two tolerances again by one order to  $5e-7$ . Solve again and find: results are the same as before. Thus, the prior tolerance  $5e-6$  worked already well.
5. Set the two tolerances back to  $5e-6$ .

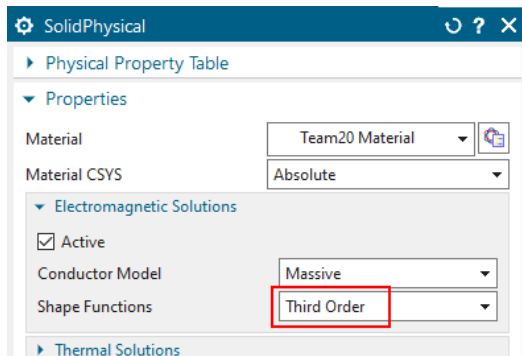
#### 4.2.12 Increasing Accuracy by global higher Order Shape Functions

The shape functions define the approximation of the main result in the finite elements. we can increase the order without changing the mesh. We use so called hierarchical shape functions, that means, no midnode elements are necessary. Increasing the order strongly increases the number of degrees of freedom (DOFs or unknowns) in the equation system. This results in more solve time and much more necessary memory (RAM). The number of DOFs is shown in the solution monitor quite at the beginning. There are several systems shown. The relevant one is simply the largest one. The picture below shows the DOFs with the origin (first order) left and a solve with third order on the right. Notice, that the number has increased by a factor of nearly 5.



To test with third instead of first order, do this:

1. switch to the Fem file,
2. edit the physicals of Core, Iron, Air (if you want, also the coil) and for each of them
3. in box 'Electromagnetic Solutions', set the 'Shape Functions' to 'Third Order'.

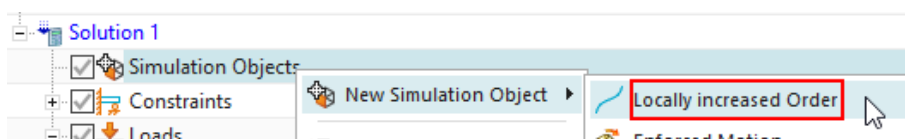


4. solve again for the three currents. The solve time will increase a lot. Compare the results:
  - 3.0 A: Measurement: -54.4 N. Simulation: -54.39 N Deviation: 0.02%
  - 4.5 A: Measurement: -75.0 N. Simulation: -74.37 N Deviation: 0.84%
  - 5.0 A: Measurement: -80.1 N. Simulation: -79.64 N Deviation: 0.57%

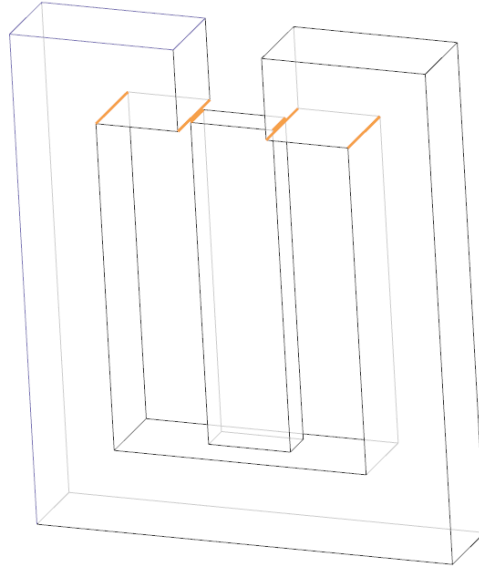
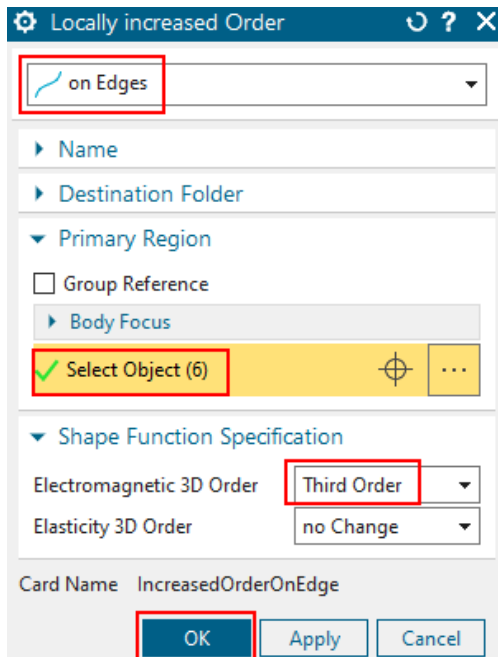
#### 4.2.13 Increasing Accuracy by local higher Order Shape Functions

Because the globally increased order of shape functions strongly increases the number of DOFs, there exists a feature for increasing that order on elements being connected to edges or faces. This is often a good compromise because the DOFs increase only slightly but the accurate shape functions are installed. For good efficiency, we should select faces or edges with sharp corners. Or simply those, where the magnetic fluxdensity changes rapidly. Proceed as follows for a test:

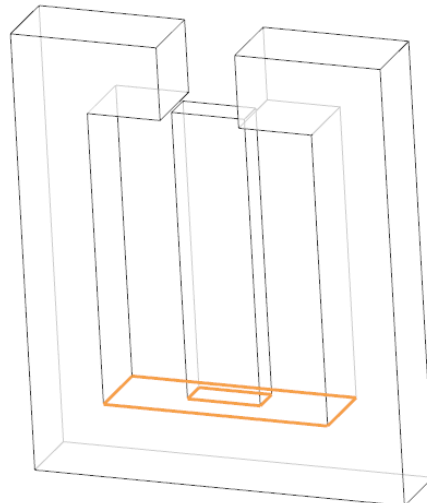
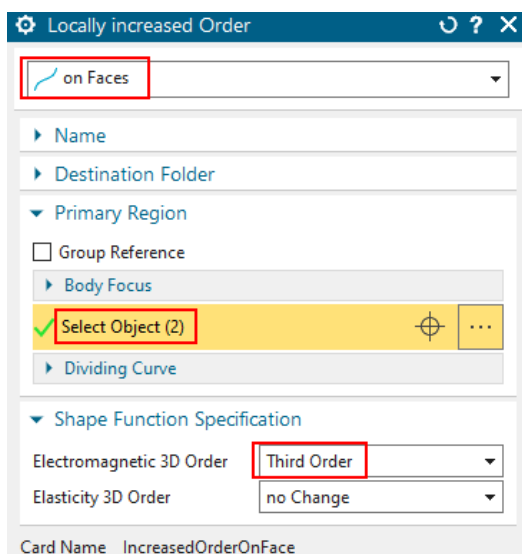
1. switch back to the Fem file and set all physical shape functions back to 'First Order'
2. switch to the Sim file.
3. Create a 'Simulation Object' of type 'Locally increased Order'



- Accept the default type 'on Edges' and select the 6 sharp corner edges at the upper side, where we have already seen the magnetic fluxdensity changing rapidly. These are shown in the picture.



- In 'Electromagnetic 3D Order', set the 'Shape Function' to 'Third Order'
- Create another 'Locally increased Order', now set the type to 'on Faces' and select the two faces at the lower side, where the flux jumps over the air-gap between the core and iron. Set the order to 3 here too.

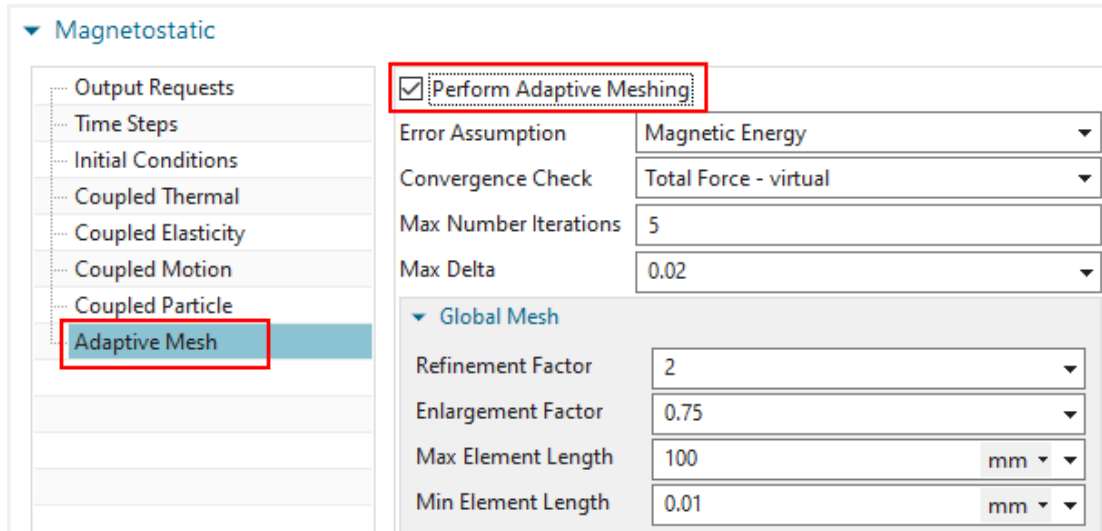


- solve again for the three currents. Notice, that we now have 53345 DOFs, only a little more than with full first order.
  - 3.0 A: Measurement: -54.4 N. Simulation: -54.96 N Deviation: 1.03%
  - 4.5 A: Measurement: -75.0 N. Simulation: -74.02 N Deviation: 1.31%
  - 5.0 A: Measurement: -80.1 N. Simulation: -78.90 N Deviation: 1.50%

#### 4.2.14 Increasing Accuracy by Adaptive Meshing

To increase the result accuracy, one can also use adaptive meshing.

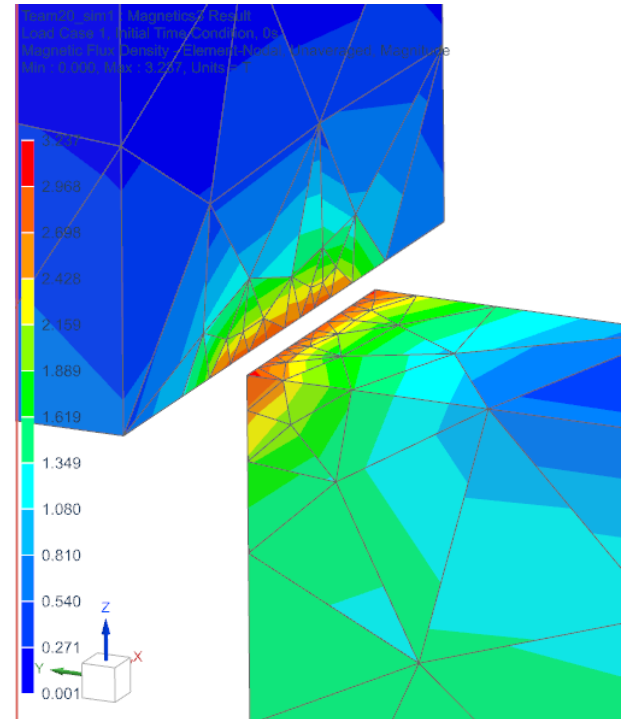
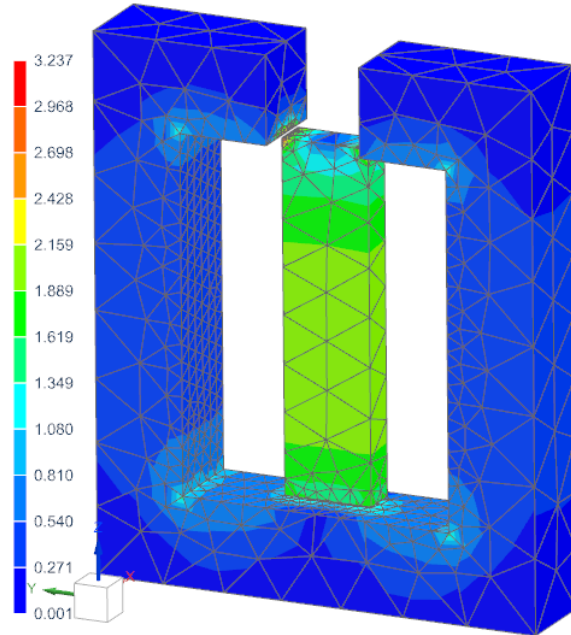
1. Remove the two 'Locally increased Order' features from the model first.
2. set the core, iron and air physicals to 'Third Order'.
3. Activate the button 'Perform Adaptive Meshing' in the edit solution dialogue at register 'Adaptive Mesh' as shown in the below picture. All default settings, as seen below, can stay in many cases.



4. The feature adaptive meshing takes more time to solve because it reflects the results and changes the mesh to find an optimal mesh. Therefore, it should be used only with one electric current value. For the other values, it should be switched off, so the system will use the already adapted mesh. Using this, we get the final result values as shown below.
  - 3.0 A: Simulation: -55.40 N Deviation: 1.8%
  - 4.5 A: Simulation: -75.99 N Deviation: 1.3%
  - 5.0 A: Simulation: -81.32 N Deviation: 1.5%

The mesh, as it comes out of the adaptive meshing process, is demonstrated in the below picture.

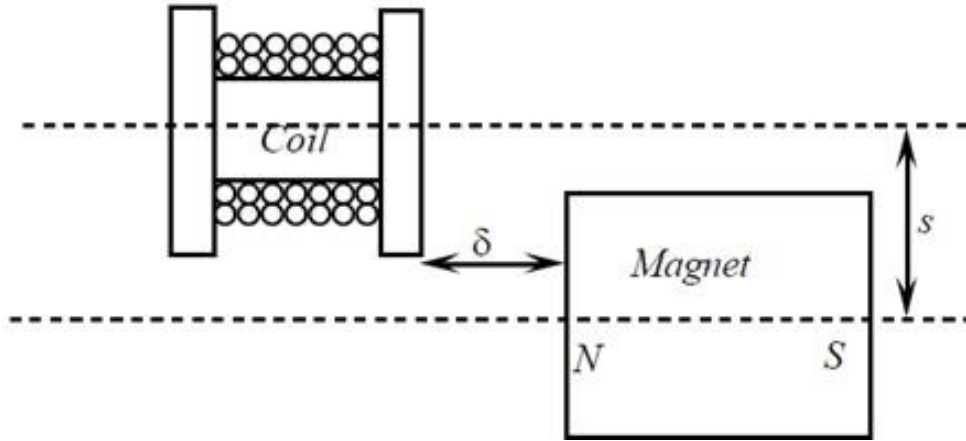
Team20\_sim1 : Magnetics3 Result  
 Load Case 1, Initial Time Condition, 0s  
 Magnetic Flux Density - Element-Nodal, Unaveraged, Magnitude  
 Min : 0.000, Max : 3.237, Units = T






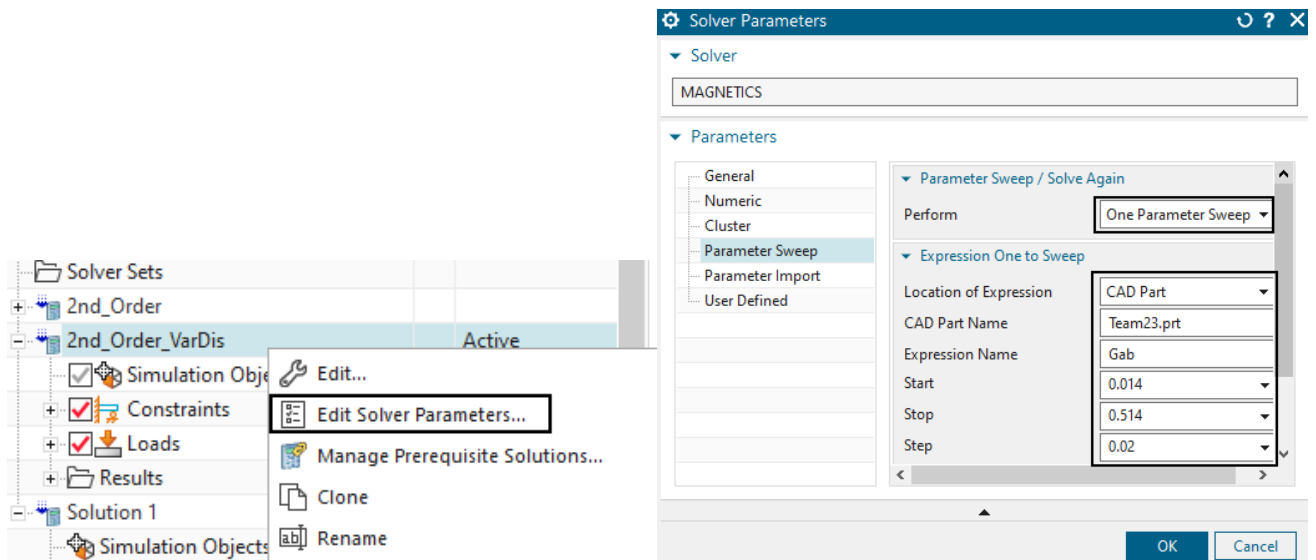
This tutorial is complete. Save your files and close them.

### 4.3 Force on Magnet (TEAM 23)

In this tutorial forces between a coil and core are computed in 2D at different positions. A parameter sweep is performed to vary different CAD positions of the magnet. The model is already set up, we just have to create a second solution with a parameter sweep.



1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/3.MagSta/3.3ForceMagnetTeam23.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program Simcenter  (or NX).
4. In Simcenter, click Open  and navigate to folder 'complete'. Select the file 'Team23\_sim1.sim' and click OK. (Maybe you must set the file filter to '.sim') This file already contains the meshes, physical properties, loads and constraints.
5. Create a new Solution .
- Key in for name '2nd\_Order\_VarDis'.
  - Accept solver 'MAGNETICS', analysis type '2D or axisym Electromagnetics' and solution type 'Magnetostatic'.
  - In 'Output Requests', 'Table', activate 'Total Force - entire (virtual)'. In '2D' activate 'Axisymmetric'.
6. click RMB on the solution '2nd\_Order\_VarDis' and click 'Edit Solver Parameters'.
  - Under 'General', switch 'Result Graphs (afu)' to 'None'.
  - Under 'Parameter Sweep', 'Perform' assign 'One Parameter Sweep'. Continue as shown in the following window.



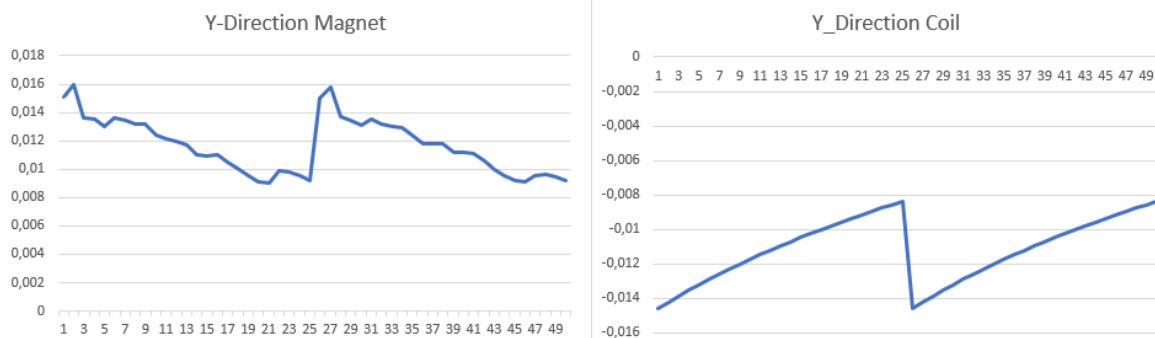
## 7. Explanation:

- The 'Location of Expression' is set to 'CAD Part' because the location of the coil body will be variable.
- Therefore we use 'Team23.prt' for 'CAD Part Name'.
- The 'Expression Name' is set to 'Gab'.
- At last, the start position, the 'Step Size' and the end position are defined.

## 8. Solve the 'VarDis' solution.

## 9. Open the '.VarDis.Force.txt' file or export the table in EXCEL.

Following two plotted graphs of the forces in the Y-direction.

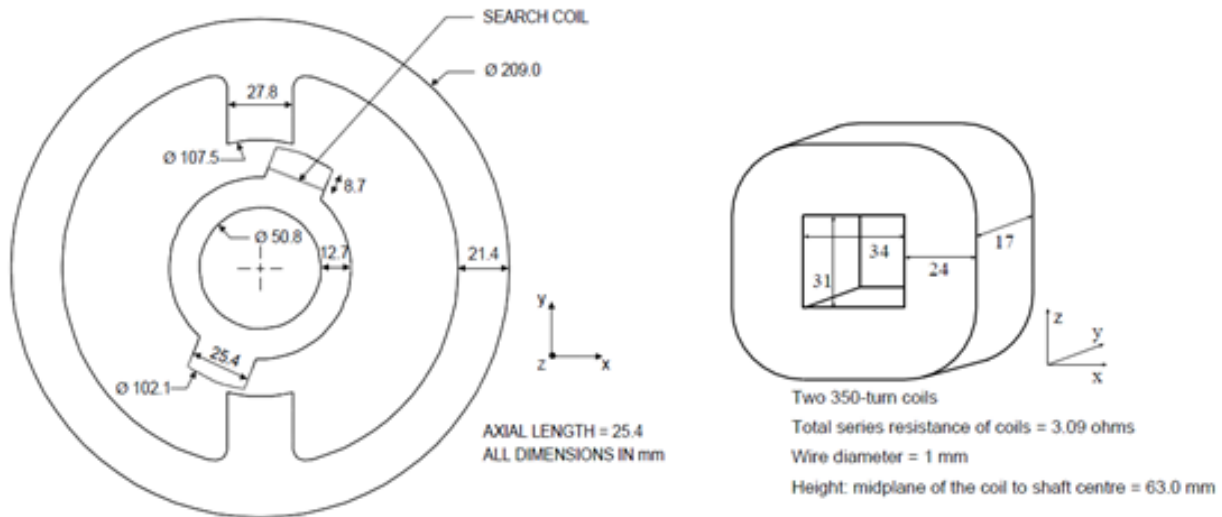


One can see that the force becomes smaller with greater distance. This is because of the magnetic effect.

## 5 Magnetodynamic

## 5.1 Nonlinear Transient Rotational Test Rig (TEAM 24)

The problem 24 is for validating 3D nonlinear time transient codes. The user can read the document 'problem24\_withResults.pdf' to get further background information for this problem.



In this problem we switch on the Voltage in two coils to a value of 23.1 V. We calculate the time dependent current in the coils, the torque and flux density over time.


Estimated time: 1.5 h





### 5.1.1 Strategy

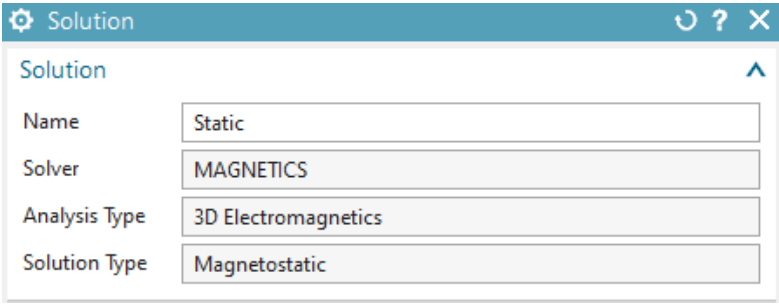
The task description does not give any information about the conductivity of the coil material. Instead it provides the ohm resistance: Both coils have 3.09 Ohm, so one will have the half of it. To account for this given resistance in our simulation there are two possibilities: We can either adjust the conductivity or the fill-factor of the coil to drive the resistance to the demanded value.

So our strategy will be building up the model using standard copper and first performing a fast static analysis to compute the ohm resistance of the coils. This first solution will be done without rotor and stator, just with the coils and air. Comparing this ohm value against the demanded one will give us a scale value. This scale value can be used to either scale the coil conductivity or the coil fill-factor. We will use the fill-factor. After this the transient run can be done.

### 5.1.2 Set up the Fem Model

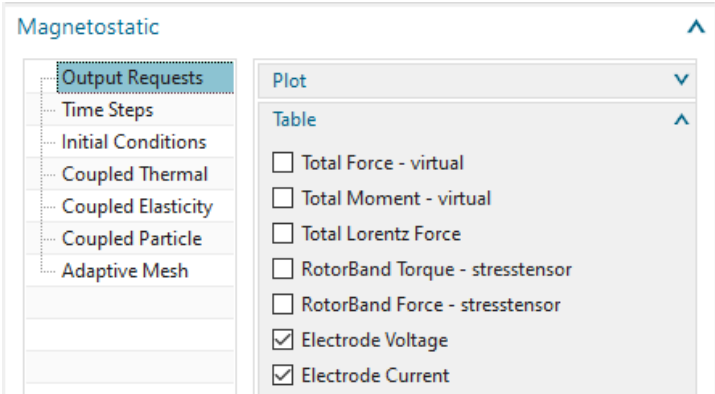
1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.2Team24.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program Simcenter  (or NX).

4. In Simcenter, click Open  and navigate to folder 'start'. Select the file 'Team24.prt' and click OK. (Maybe you must set the file filter to 'prt')
5. From toolbar Application click on 'Pre/Post' 
  - In toolbar 'Magnetics', create a 'New Fem and Sim File (Non-Manifold)' ,  
Hint: If NX Simcenter version is older than 2212, this option is not available. See tutorial Team20 (beginner) for alternative methods.
6. Create a first solution  of type 'Magnetostatics'. This will be the pre-solution.
  - Name the solution 'Static'
  - Accept solver 'MAGNETICS' and
  - analysis type '3D Electromagnetics',




Solution	
Name	Static
Solver	MAGNETICS
Analysis Type	3D Electromagnetics
Solution Type	Magnetostatic

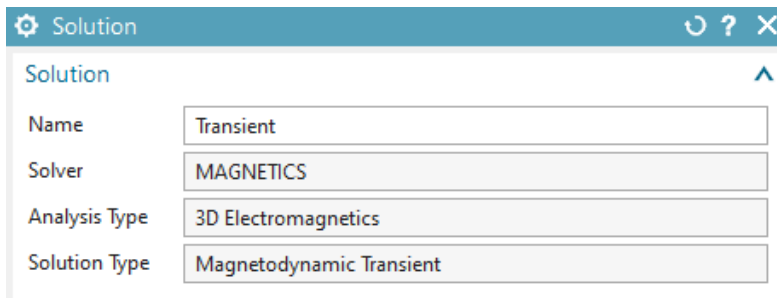
- In the output requests, activate the tabular results 'Electrode Voltage' and 'Electrode Current' to get tabular output for voltage and current on the coils. This will allow us to compute the ohm resistance by the formula  $R = U/I$ .



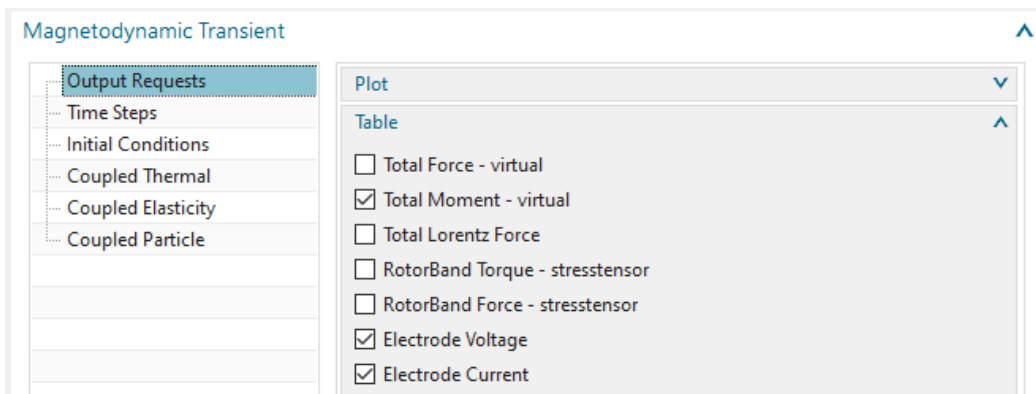
Magnetostatic	
<b>Output Requests</b> Time Steps Initial Conditions Coupled Thermal Coupled Elasticity Coupled Particle Adaptive Mesh	<b>Plot</b> <b>Table</b> <input type="checkbox"/> Total Force - virtual <input type="checkbox"/> Total Moment - virtual <input type="checkbox"/> Total Lorentz Force <input type="checkbox"/> RotorBand Torque - stresstensor <input type="checkbox"/> RotorBand Force - stresstensor <input checked="" type="checkbox"/> Electrode Voltage <input checked="" type="checkbox"/> Electrode Current

- Click OK.

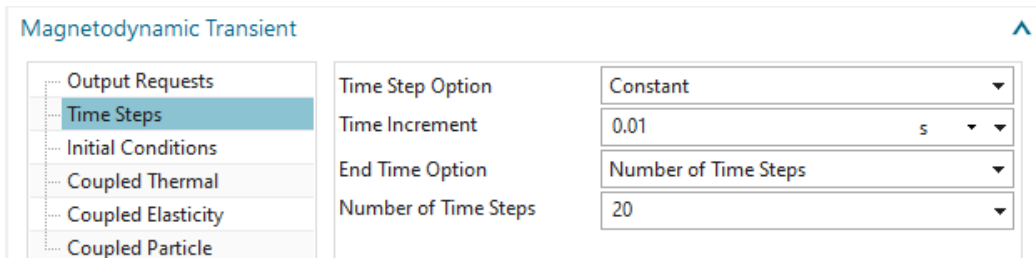
7. Create a second solution (in the Sim part, use button ) of type 'Magnetodynamic Transient'.



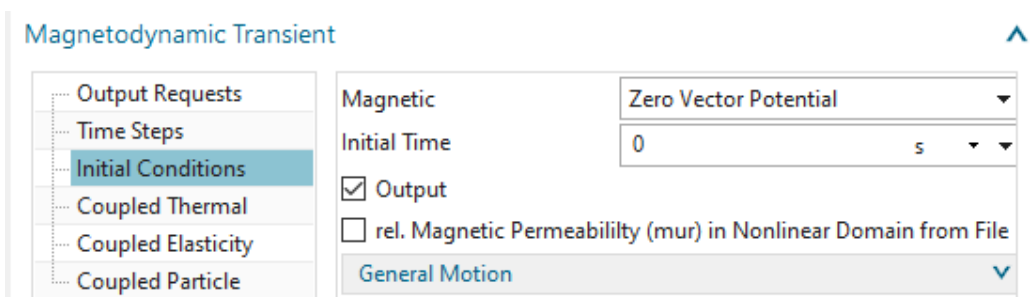
- Name the solution 'Transient'
- In the output requests 'Plot', activate 'Magnetic Flux Density' and 'Current Density'. In the 'Table', activate 'Total Moment - entire (virtual)' to get tabular output of rotor torque over time steps. Also activate 'Electrode Voltage' and 'Electrode Current'. Feel free to activate more results as desired.



- In register 'Time Steps', set 'Number of Time Steps' to 20 and 'Time Increment' to 0.01 sec.



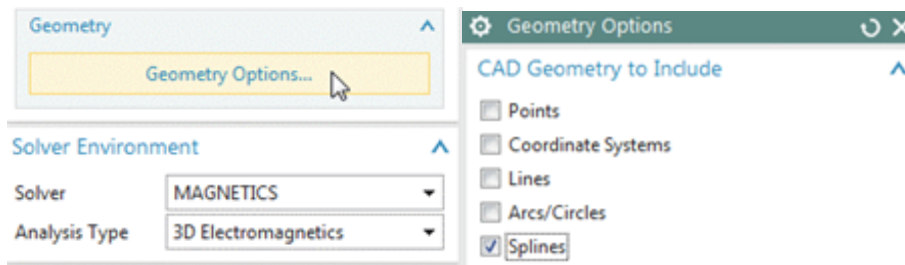
- In register 'Initial Conditions', activate 'Output' to output the results of time zero.



- Click OK.

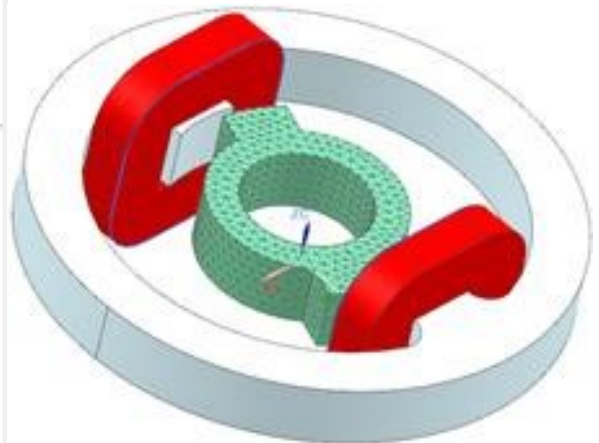
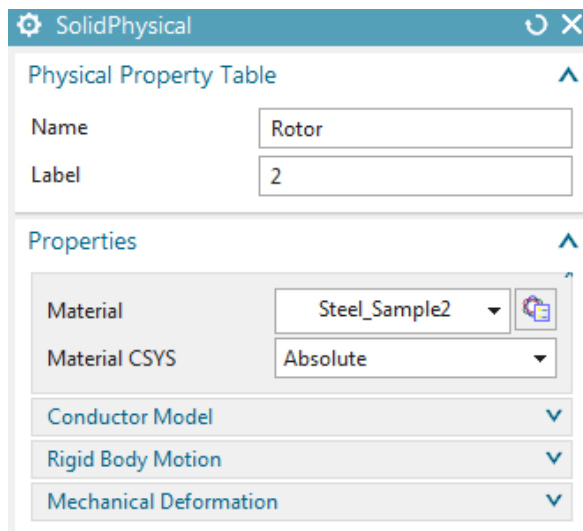
8. Set the Fem file to the displayed part.

9. Edit the Fem file and activate 'Splines' in Geometry Options.



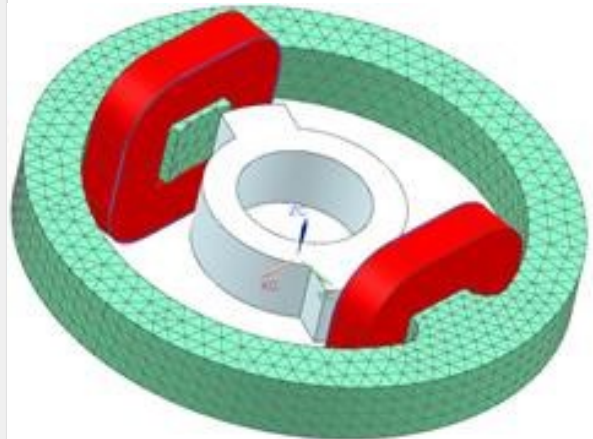
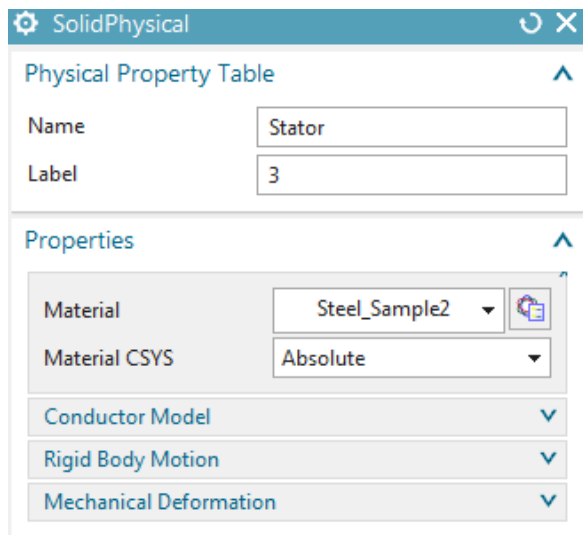
10. The Rotor

- Mesh using tetrahedral elements with the half of the suggested element size ( $8.83/2$  mm).
- Apply material 'Steel\_Sample2' (From library 'Magnetics\_Materials.xml' in folder MAGNETICS). Notice that this material has a nonlinear B-H curve. It contains the data of the TEAM24 task description.



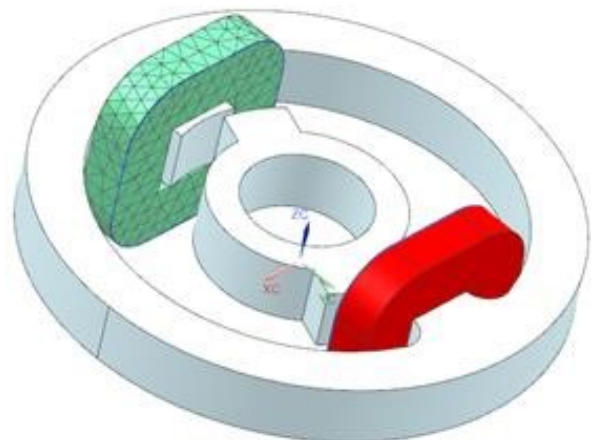
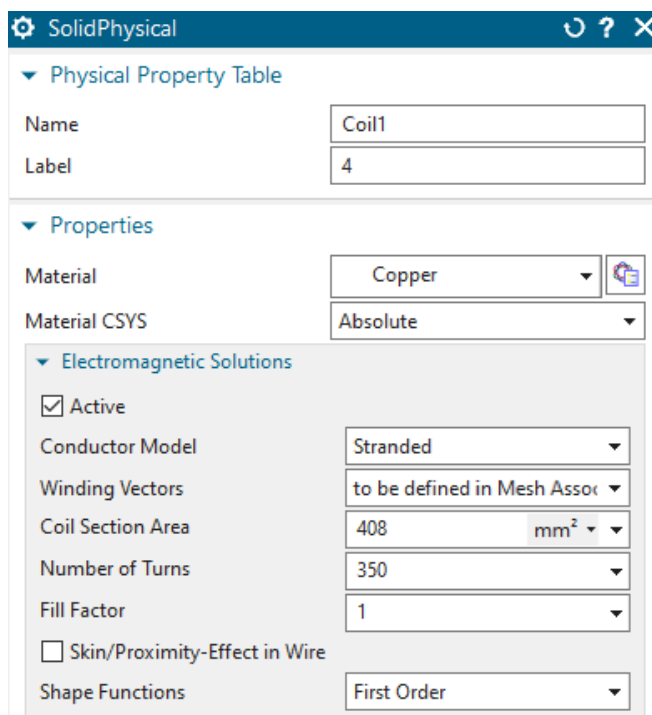
11. The Stator

- Mesh with tets and half of suggested element size ( $14.6/2$  mm).
- Apply material Steel\_Sample2.



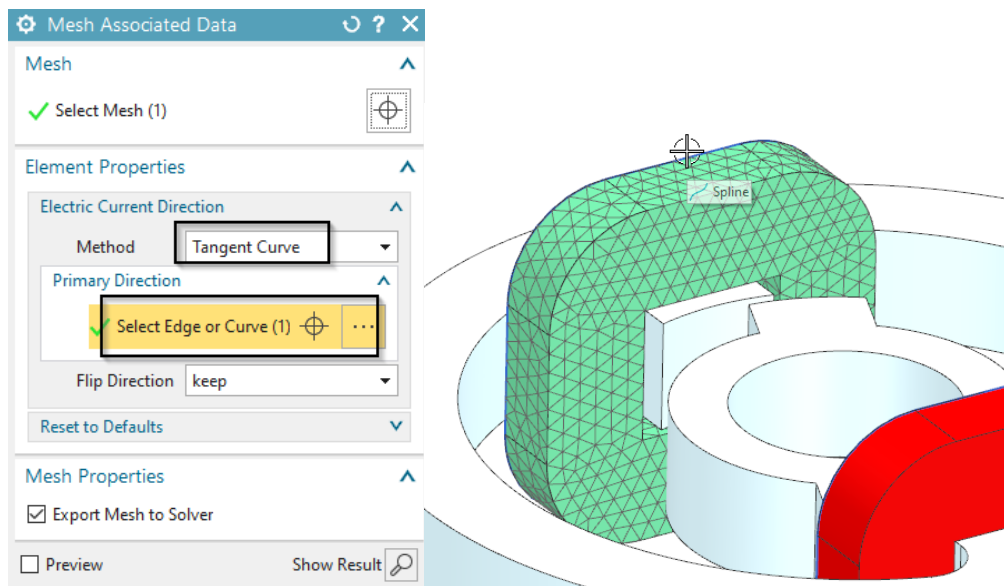
## 12. The first Coil

- Mesh by tets and the suggested element size (8.92 mm).
- Name the physical 'Coil1'.
- Set the material to 'Copper'. Make sure this copper has a electric conductivity (58e6 S/m).
- Set the 'Conductor Model' to 'Stranded', use 350 Turns and a Section Area of 408  $\text{mm}^2$ .
- Accept the default 'Fillfactor' of 1. Later we will adapt this to drive the coil to the demanded resistivity value.

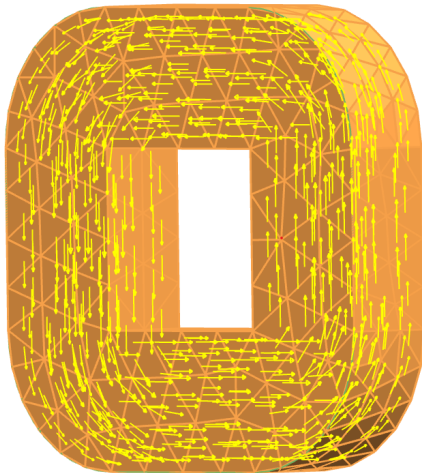


- Edit the 'Mesh Associated Data' of Coil1 mesh. Set the 'Electric Current Direction' option to 'Tangent Curve' and select the spline curve to define the direction of the

windings.



- Hint: Activate 'Preview' to check the winding direction. It should look like the picture below.

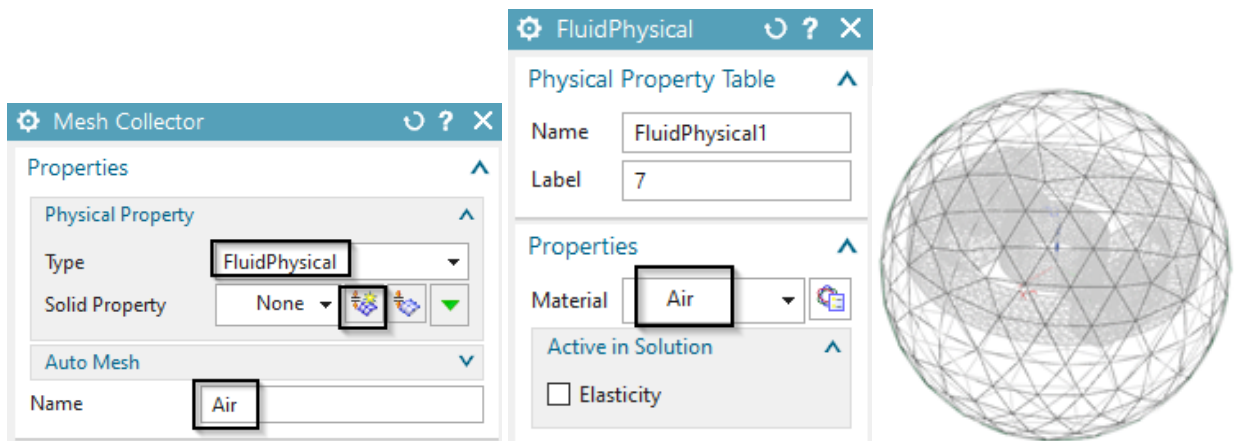



13. The second coil

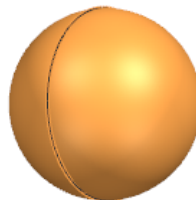
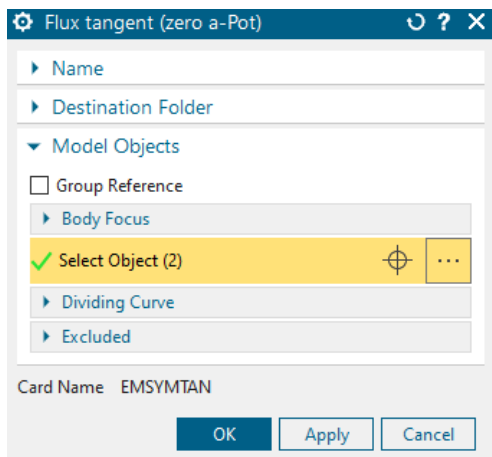
- Use the same steps as for the first coil. Assign 'Coil2' for name.

14. The Air:

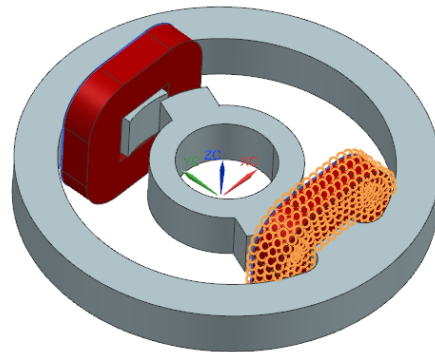
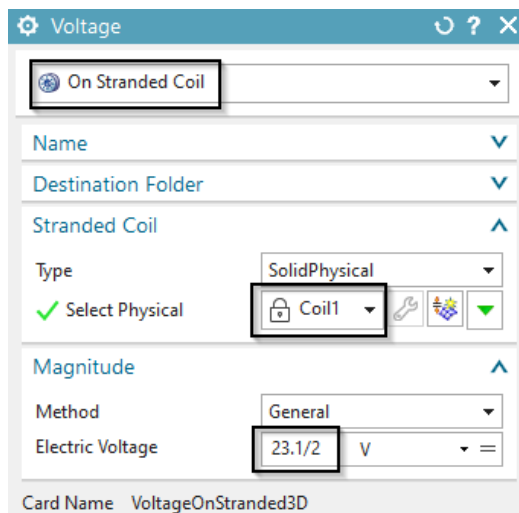
- Mesh the air volume with tets and use the suggested element size (34.5 mm).
- Set the physical to 'FluidPhysical' and assign material Air.



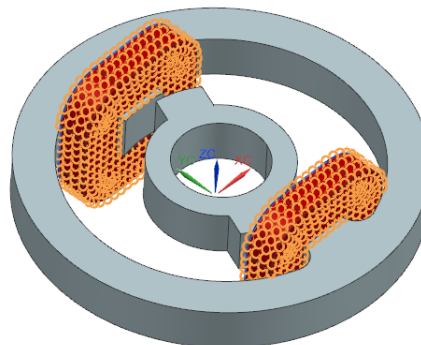
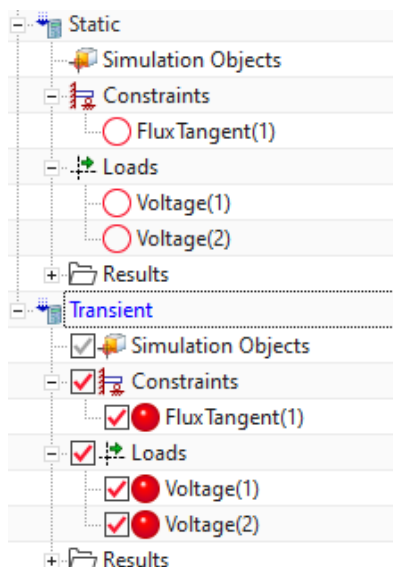
15. For easier post processing, click on the button 'Rename Meshes ...'  from the Magnetics toolbar. That utility renames all meshes and therefore is a useful feature especially with larger models and many meshes.
16. Switch to the Sim file.
17. Create a constraint of type 'Flux tangent (zero a-Pot)' on the sphere outside faces.



18. Create the voltage loads.
  - Apply voltage load of type 'On Stranded Coil' on the first coil. Assign 23.1/2 V.

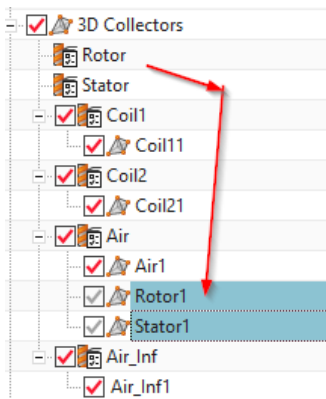


- Apply a second voltage on the second coil with the same value.
- Assign these two loads and the constraint (by drag and drop) to both solutions. The navigator should look like in the picture.



19. Prior to solving the static solution we must avoid that the coils are influenced by the rotor and stator. Therefore,

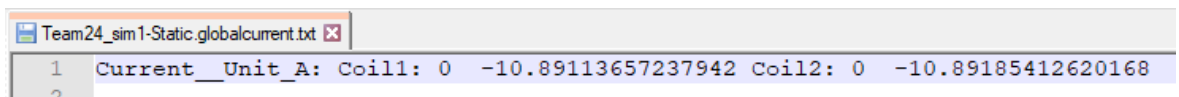
- change to the Fem part,
- drag and drop the meshes of the rotor and stator into the air collector. Now these two will behave as air and not affect the coils,



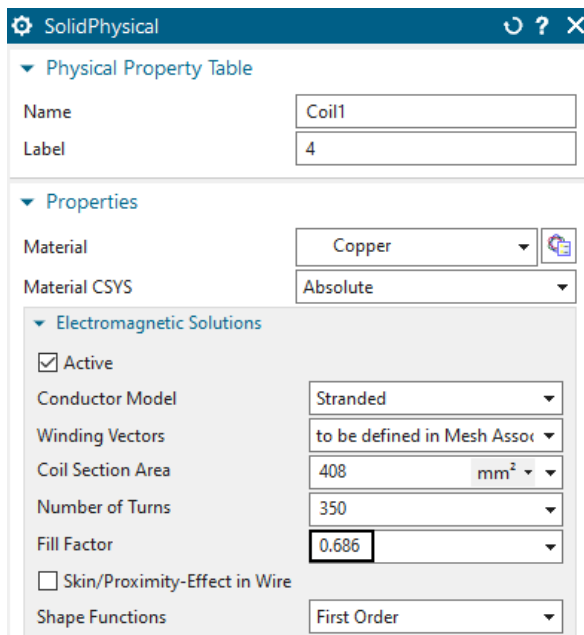
- change back to the Sim part and solve the 'Static' solution. The solution time should be about 0.5 minutes.

### 5.1.3 Find Coil Fillfactor, Rerun and Test

- After the solve job has finished open the newly created afu- or text file Team24\_sim1-Static.globalcurrent.txt with the XY Function Navigator or with a text editor. Find there the resulting current at the two coil electrodes. The result is about 10.89 A.



- According to  $R = U/I$  we can compute the actual resistance to  $R = 11.55V/10.89A = 1.06\Omega$ . Because the demanded resistance is  $3.09/2\Omega = 1.545\Omega$  the scale factor becomes  $1.06/1.545 = 0.686$ . To use this factor as fillfactor switch to the Fem part and edit the two physicals of the two coils. Set the 'Fillfactor' as shown in the picture:

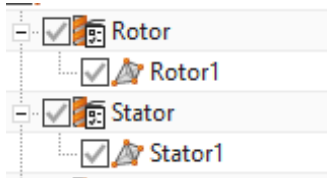


- Test solve again the static solution and find the new current result as 7.47 A.

```
Team24_sim1-Static.globalcurrent.txt
1 Current__Unit_A: Coil1: 0 -7.471319688652319 Coil2: 0 -7.47181193057432
2
```

- Now the resistance of the coil is  $R = 11.55 \text{ V} / 7.47 \text{ A} = 1.54 \text{ Ohm}$  as it is demanded in the Team task description.

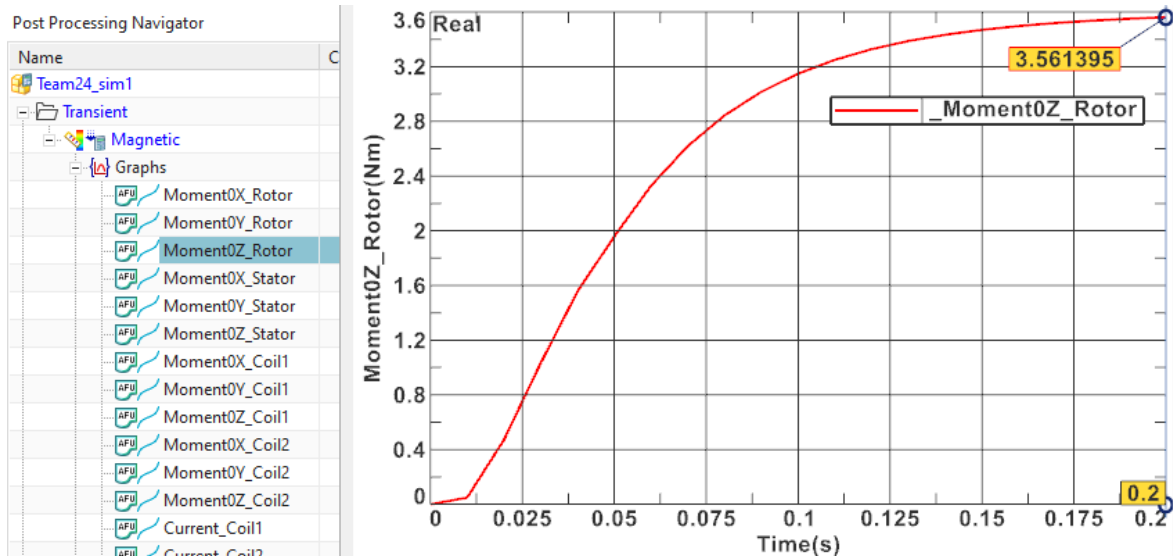
20. Prior to solving the transient solution, move the two meshes of rotor and stator back into their origin mesh collectors. Then solve. The solution time will be about 3 minutes because of nonlinear iterations due to saturation effects.



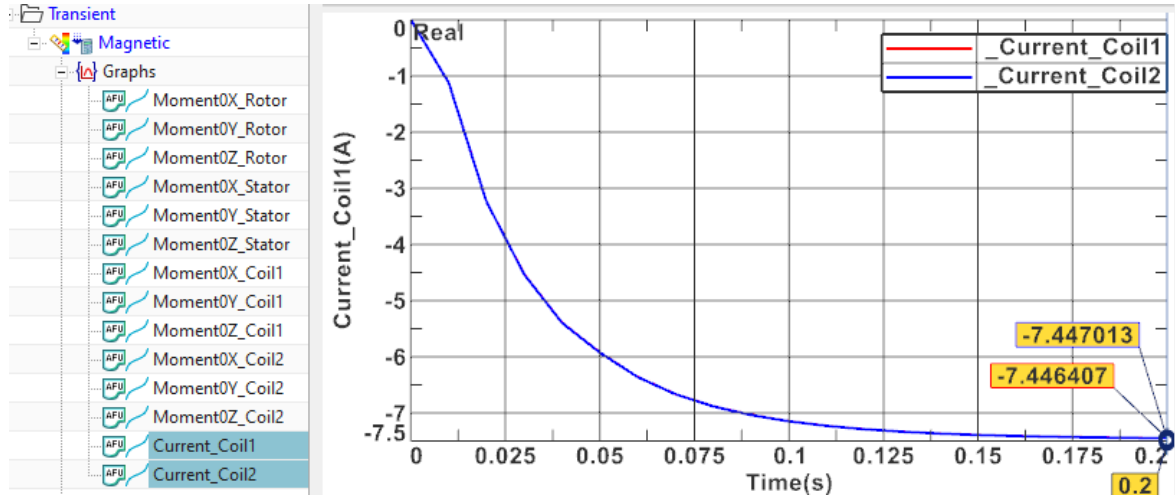
### 5.1.4 Post Processing

Check current and torque results over time.

1. Open the graphical table results and display them as follows.
2. Display z component of the torque on the rotor. The maximum value is 3.56 Nm. The reference result from measurements is 3.2 Nm. Smaller mesh sizes will drive the simulation result more and more against the measurement.



3. Display the current result for both coils. They show a good agreement and the maximum value is 7.45 A. The measured reference value for the current is 7.4 A at time 0.2 sec.



A manual verification of the time dependent current  $I(t)$  in a coil after switching on the voltage, is given by the formula:

$$I(t) = (1 - e^{\frac{R}{L} \cdot t}) \cdot \frac{U_0}{R}$$

with  $R = 1.54\Omega$ ,  $L = 0.03H$ ,  $U_0 = 11.55V$  and  $t = 0.01s, \dots, 0.2s$ .  
 $L$  is precalculated by the formula:

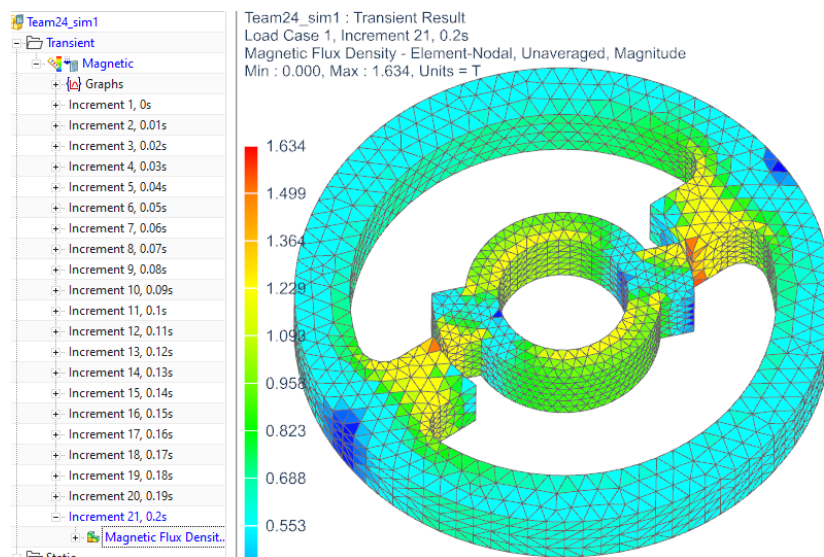
$$L \approx \mu_r \cdot \mu_0 \cdot N^2 \cdot \frac{A}{l} \approx 0.03H$$

Following a table with a comparison of the simulation and analytic results:

Time	Simulation Result	Analytic Result	Deviation
0.01s	2.65A	3.01A	13.6%
0.05	6.2A	6.92A	11.6%
0.1s	7.2A	7.45A	3.5%
0.15s	7.38A	7.49A	1.5%
0.2s	7.43A	7.4997A	0.9%

The values match well at the end time. Deviations in the beginning are higher probably because of the unsecure inductivity value of the coils.

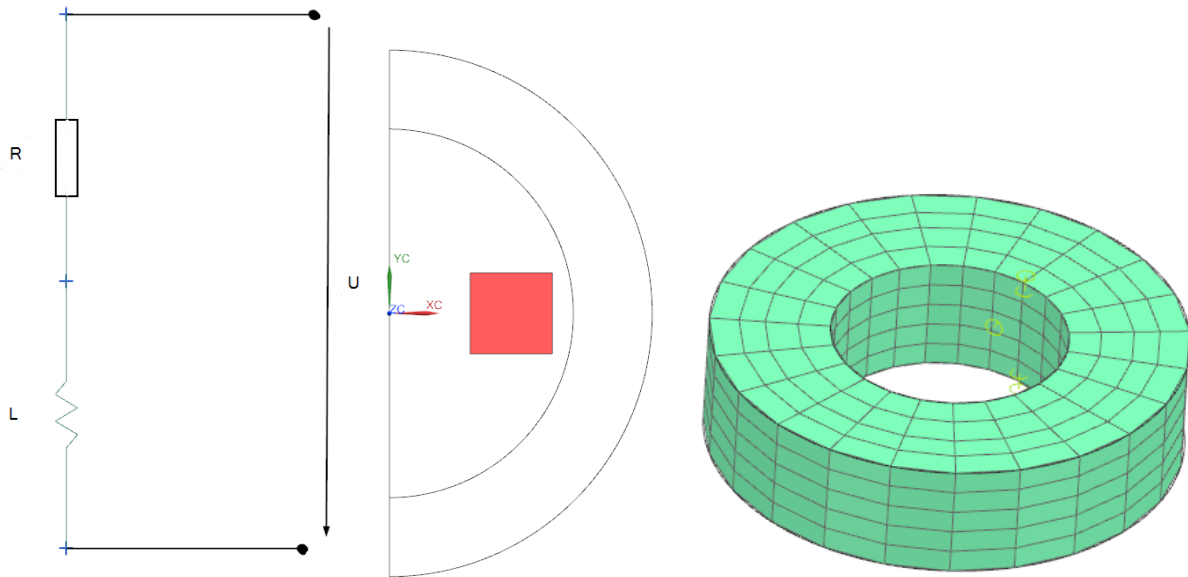
4. The contour plot of the flux density at time 0.2 s (last time step) looks like this:



The tutorial is finished.


## 5.2 Inductance, Phase Shift and Resistance of a Coil in 1D,2D,3D

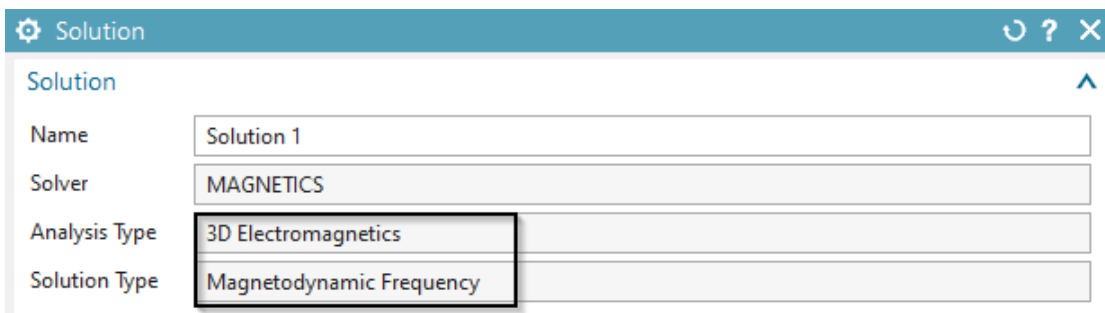
In this example we show how to calculate inductance, ohm resistance, voltage, current and their phase shift in a simple coil. We start with a 1D simulation where we assign the values of current, inductivity and resistance. The simulation calculates the resulting voltage. In a similar way one could also set up much more complex electrical circuits and compute for such results. Following we simulate this in a 2D axisymmetric and also a 3D simulation. The advantage of 2D and 3D simulation is that the parameters of the coil are calculated with the given geometry by FEM. Another possibility is to combine 1D circuits with 2D and 3D models, what is not shown in this tutorial.



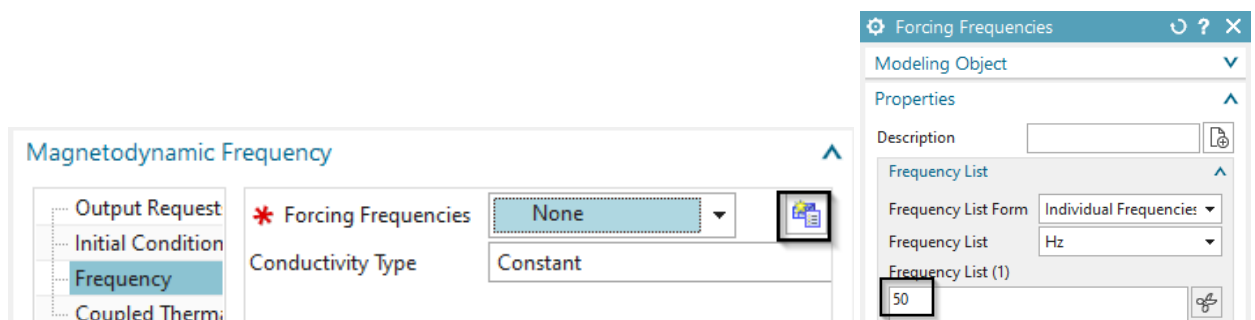
Estimated time: 1 h.

### 5.2.1 1D Circuit Simulation

1. Start the Program Simcenter  (or NX). Use Version 10 or higher.
2. Create a new .prt file.
3. Start the application Pre/Post.
4. Choose solver 'MAGNETICS' and Analysis Type '3D Electromagnetics' ('2D or axisym Electromagnetics' is also possible). Click OK.
  - Set Solution Type to 'Magnetodynamic Frequency'.



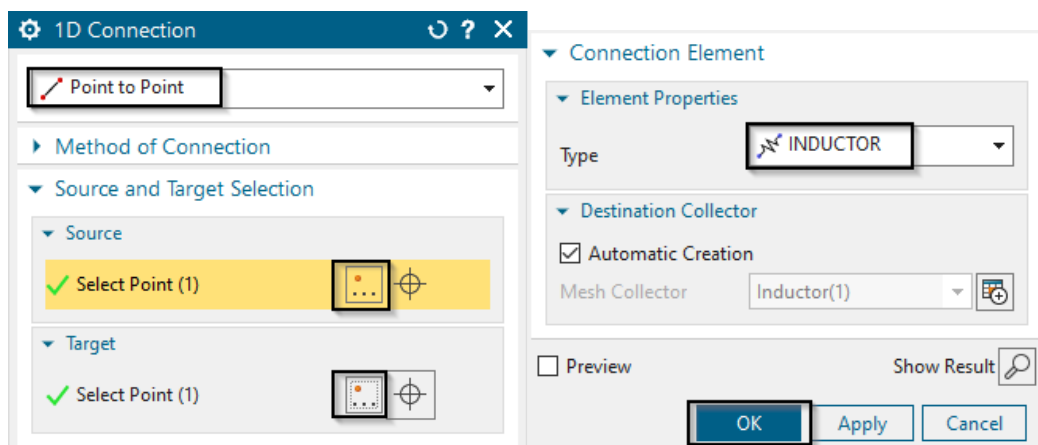
- In register 'Output Requests' under 'Table' activate 'Circuit Voltage'.
- In register 'Frequency Domain', create a modeling object and set the 'Forcing Frequency' to 50 Hz(Picture below).



5. Switch to the Fem-file.

6. From toolbar 'Home' create a '1D Connection'

- Switch from 'Node to Node' to 'Point to Point'
- In 'Source' open the 'Point Dialog' and key in '100' for the X coordinate. Accept '0' for Y and Z. Do the same in 'Target' with  $X = 100$ ,  $Y = 100$ ,  $Z = 0$ . (Any other coordinates are possible)
- Under 'Element Properties' switch to 'Inductor'.



- Click OK. There now appears a symbol of the inductor between the two points.

7. In the Simulation Navigator double click on the created mesh collector 'Inductor(1)'.

- Click edit.
- In 'Normal Values' key in '1' H.

8. Create again a 1D Connection.

- As 'Source' use the first Point. As 'Target' create a new point with  $X = 100$ ,  $Y = 200$ ,  $Z = 0$ .
- Switch the type of the 'Connection Element' to 'Resistor'. Click OK. The symbol of the resistor is a straight line.

9. Edit the properties of the resistor. Type '1'  $\Omega$  in 'Electric Resistance per Element'.
10. Switch to the Sim-file.
11. Create a new Load. Use 'Current Harmonic'.
  - Switch from 'On Stranded Coil' to 'On Circuit'
  - For the 'Primary Node' select the point at the bottom of the inductor. For the 'Secondary Node' select the point on top of the resistor.
  - The 'Electric Current Amplitude' is set to '3'A.
12. Click RMB on the Solution, select 'Edit Solver Parameters' and under 'Result Graphs(afu)' switch to 'Create, keep txt Files'. Solve the Solution.

### 5.2.2 Post Processing

Postprocessing is done by some text or afu files that contain the results.

1. Open the file with extension \*.VoltageCircuits.txt. The Solution is divided in the applied voltage, the 'Inductor Element' and the 'Resistor Element'. Each has three parameters. The first is the frequency following the real part and last the imaginary part.

```
line_designer_model1_sim1-Solution_1.VoltageCircuits.txt
1 VoltageCircuits__Unit_V Current: 50 3 942.477796076938 Inductor: 50 0 942.477796076938 Resistor: 50 3 0
```

- Applied voltage: real part = 3V, imaginary part = 942.48 V.
- Inductor: real part = 0 V, imaginary part = 942.48 V.
- Resistor: real part = 3 V, imaginary part = 0 V.
- Impedance of the inductor:  $Z_L = U/I = (0V + j942.48V)/3A$   
 Impedance of the resistor:  $Z_R = U/I = (3V + j0V)/3A$   
 Impedance of circuit :  $Z_{RL} = Z_L + Z_R = 1\Omega + j100\pi\Omega$

### 5.2.3 Analytic Verification

For the calculation of the voltage in the coil we need the 'reactive' resistance of the coil  $X_L$ . This is calculated by:

$$X_L = j\omega L = j \cdot 2\pi \cdot f \cdot L = j100\pi$$

with  $f = 50Hz$  forcing frequency,  $L = 1H$  inductivity of coil.



To calculate the voltage we use  $U = X_L \cdot I = j100\pi \cdot 3A = 942.478V$

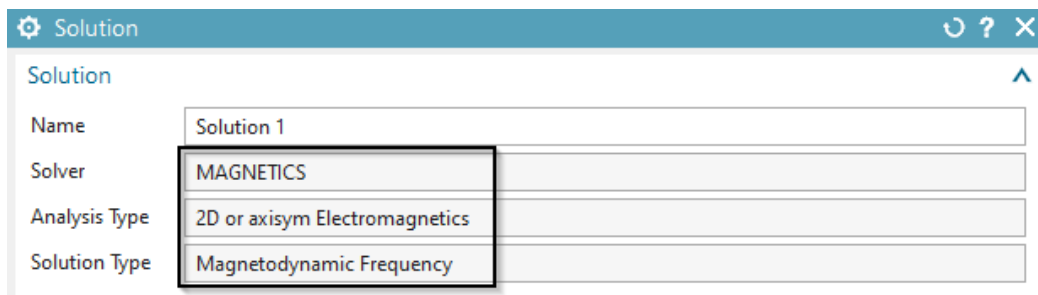
The resistance of the resistor was set to  $R = 1\Omega$ .

So the voltage in the resistor is  $U = R \cdot I = 1\Omega \cdot 3A = 3V$

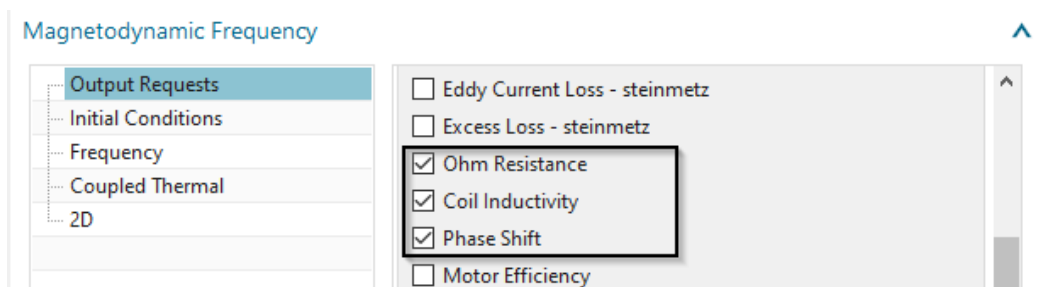
These results match with the results of the simulation.

### 5.2.4 2D Axisymmetric Analysis

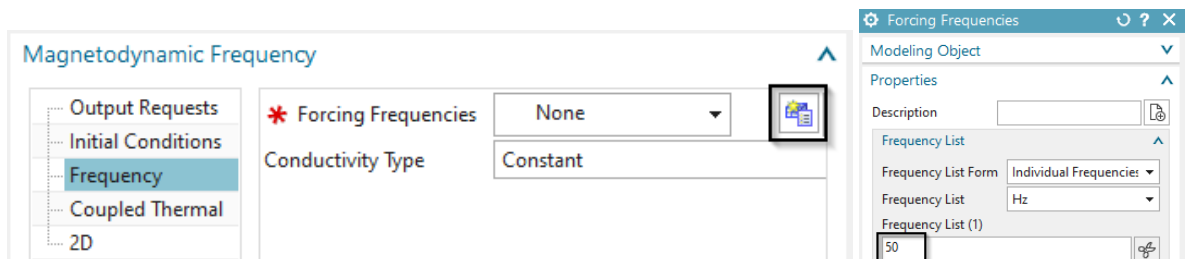
1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.4Coil.zip>
  2. unzip the archive. There will be one folder 'start' and one 'complete'.
  3. Start the Program Simcenter  (or NX). Use Version 10 or higher.
  4. In Simcenter, click Open  and navigate to folder 'start'. Select the file 'coil2D.prt' and click OK. (Maybe you must set the file filter to 'prt')
  5. Start the application Pre/Post.
  6. Create a New Fem and Sim. Set the solver to 'MAGNETICS' and Analysis Type '2D or axisym Electromagnetics'. Click OK.
  7. A new window appears asking for the solution type. See below.
- Set the Solution Type to 'Magnetodynamic Frequency'.



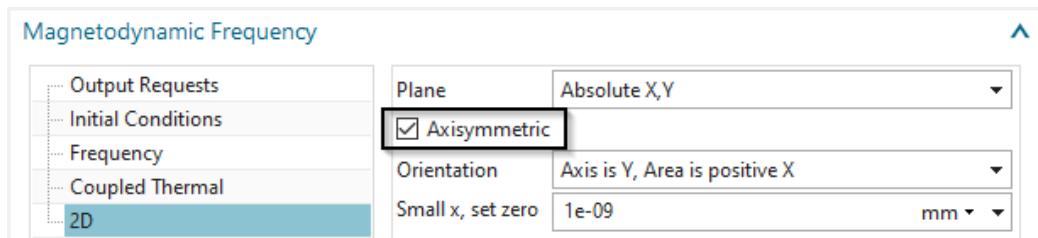
- In register 'Output Requests', 'Table' activate 'Ohm Resistance', 'Coil Inductivity' and 'Phase Shift',



- In register 'Frequency' create a modeling object for 'Forcing frequencies' with 50 Hz.

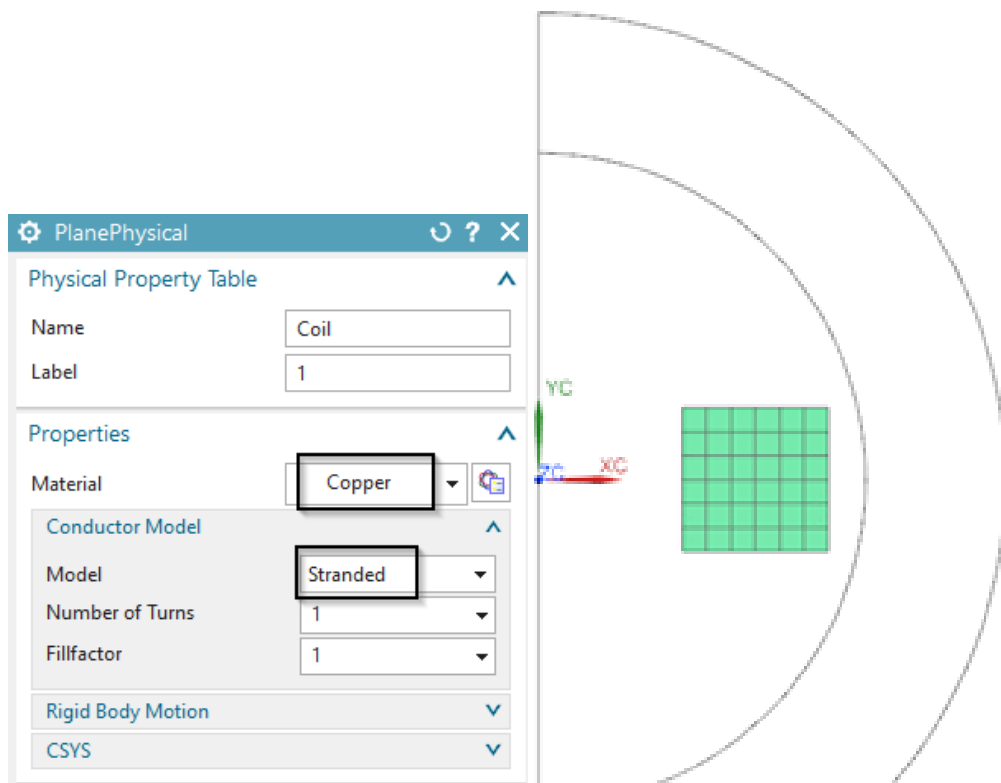


- In register '2D' activate 'Axisymmetric' and click OK.

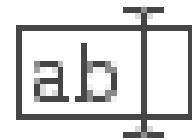
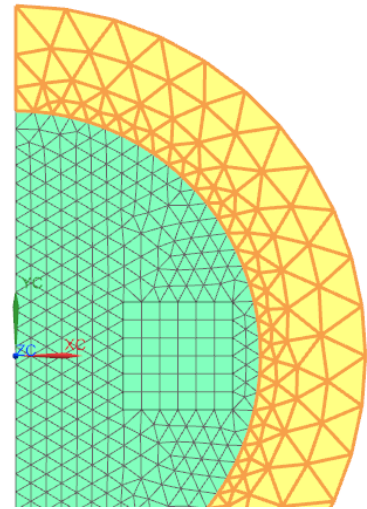
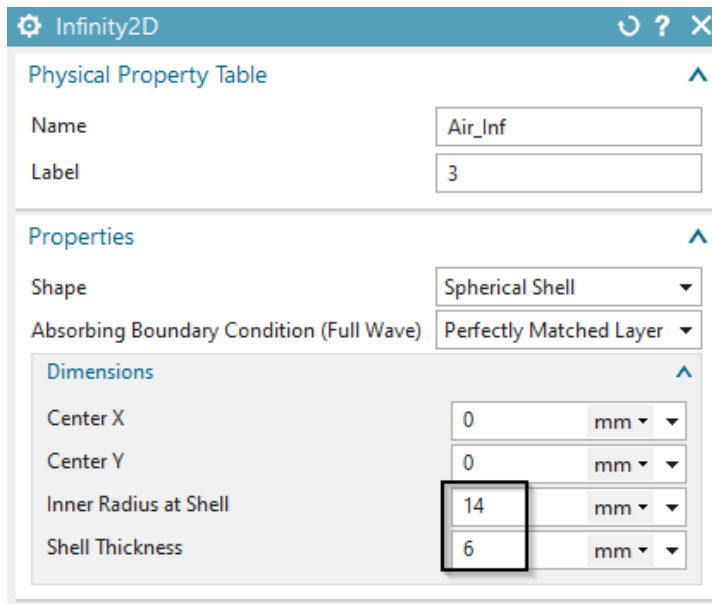


8. Switch to the FEM file.

- Mesh the conductor face using quad elements and the suggested element size /2.
- Assign 'Copper' material and set the 'Conductor Model' to 'Stranded'.



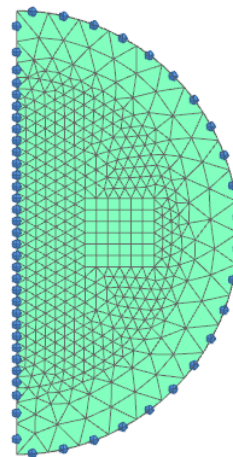
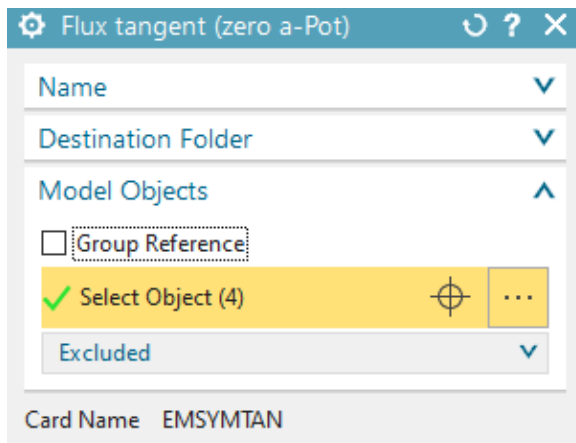
- Mesh the air face using tri elements (half of suggested size) and assign 'Air'.
- Mesh the Infinity area with tri elements (suggested element size). Assign a physical of type 'Infinity2D' and use the dimensions as shown in the picture.



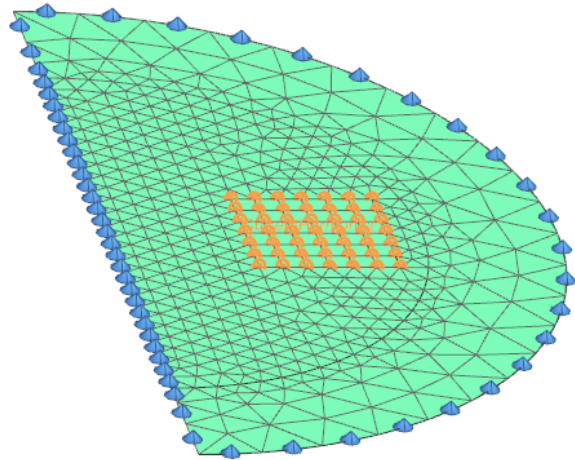
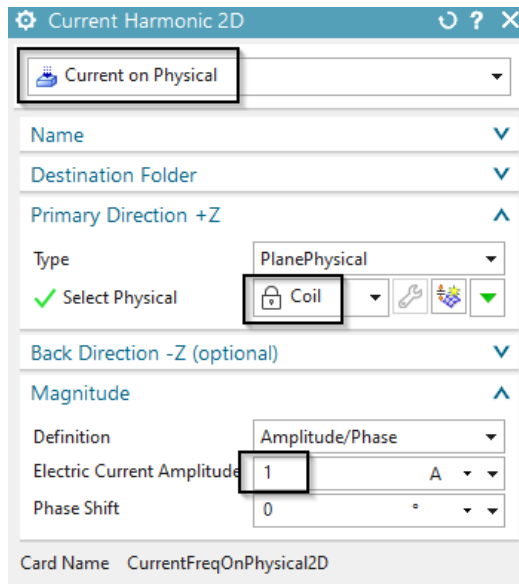
9. For easier post processing, click on the button 'Rename Meshes ...' from the Magnetics toolbar. That utility renames all meshes and therefore is a useful feature especially with larger models and many meshes.

10. Switch to the Sim file.

- Create a Constraint of type 'Flux tangent (zero a-Pot)' at the middle axis and on the circular border.



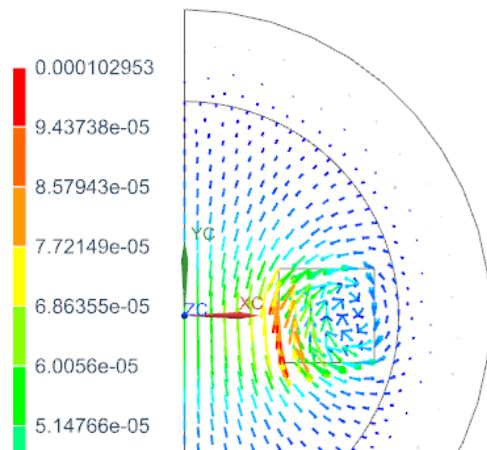
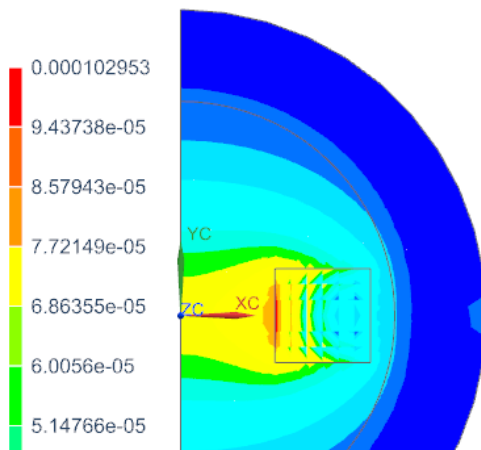
- Create a Load of type 'Current Harmonic 2D' with type 'Current' on the conductor face. Assign an Electric Current of 1 A and a Phase Shift of zero deg.



11. Under Solution, select 'Edit Solver Parameters' and modify the field Result Graphs (afu) to 'Create, keep txt Files'.
12. Solve the solution.
13. Check the flux density result. (real part)

Coil2d\_sim1 : Solution 1 Result  
 Load Case 1, Frequency 1, 50Hz  
 Magnetic Flux Density - Element-Nodal, Unaveraged, Magnitude  
 Complex Option : At Phase Angle 0.0  
 Min : 6.99376e-24, Max : 0.000102953, Units = T

Coil2d\_sim1 : Solution 1 Result  
 Load Case 1, Frequency 1, 50Hz  
 Magnetic Flux Density - Element-Nodal, Unaveraged, Magnitude  
 Complex Option : At Phase Angle 0.0  
 Min : 0, Max : 0.000102953, Units = T



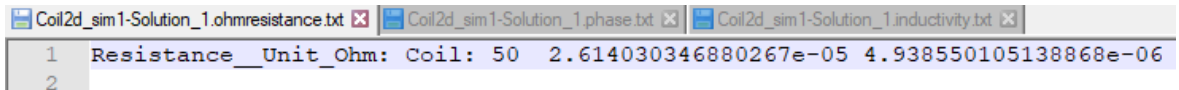
## 5.2.5 Post Processing

Postprocessing is done by some text or afu files that contain the results.

### 1. Ohm-Resistance Result

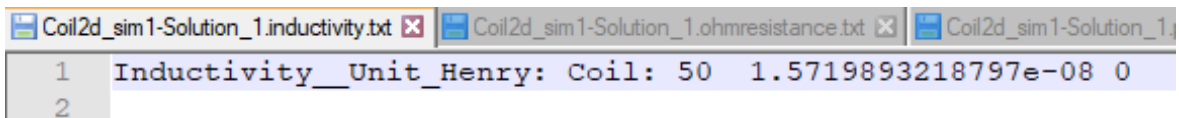
- Open the file with extension \*.ohmresistance.txt. There are two results for each region. The first one is the 'Active Resistance' (Wirkwiderstand) and the second is the 'Reactive Resistance' (Blindwiderstand). The resistance is derived from the voltage and current results by  $R = U/I$ . On region 'Current(1)' the result is

- 'Active Resistance':  $2.6\text{e}-5$  Ohm
- 'Reactive Resistance':  $4.9\text{e}-6$  Ohm .
- This result is valid for a coil with one winding. For a coil with N windings you can set the 'Number of Turns' in the conductor properties to the desired N. Alternatively the result values can be multiplied by  $N^2$ .



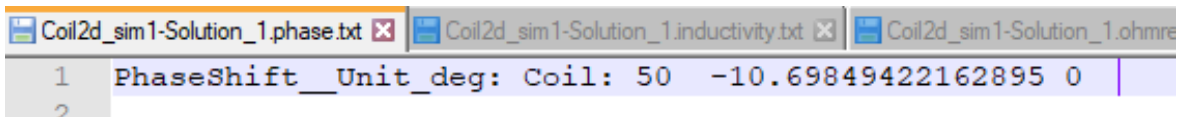
## 2. Inductance Result

- Open \*.inductivity.txt.
- The Inductance is derived from the voltage and current results by  $L = (U/I)/(2\pi f)$ . On region 'Coil' the result is  $1.57\text{e}-9$  Henry. Again this result is valid for a coil with one winding. For a coil with N windings the result value has to be multiplied by  $N^2$ .



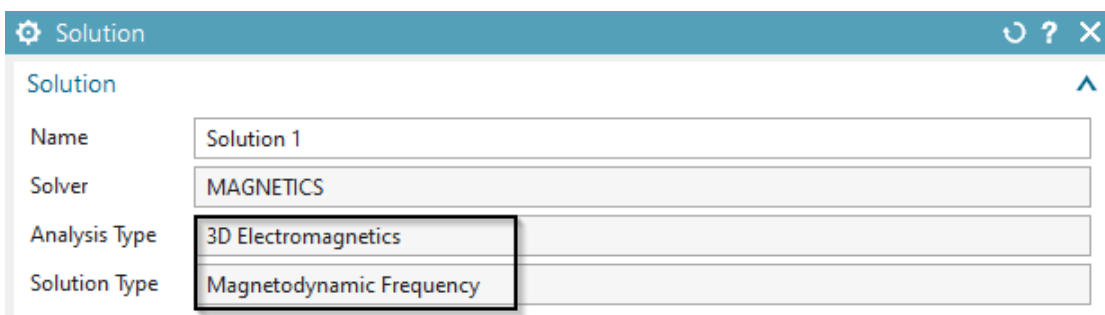
## 3. Phase Shift Result

- Open \*.phase.txt.
- The phase shift is derived from voltage results by  $\Phi = \arctan(U_{imag}/U_{real})$ . On region 'Coil' the calculated phase shift is 6 deg. This result does not depend on the number of windings.

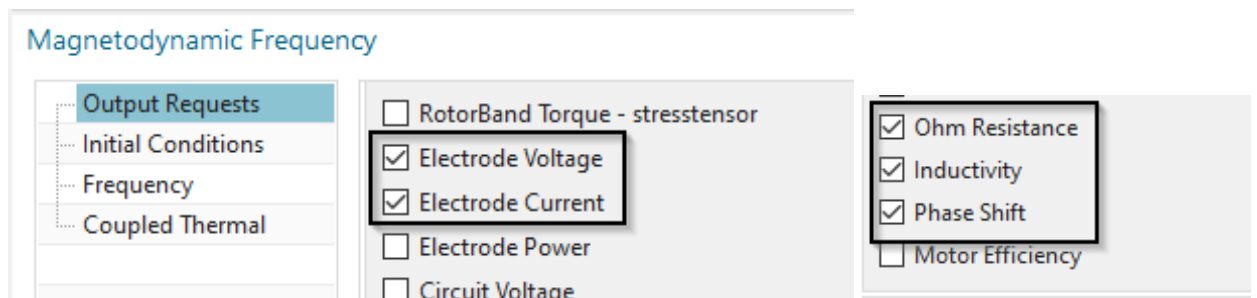


## 5.2.6 3D Analysis

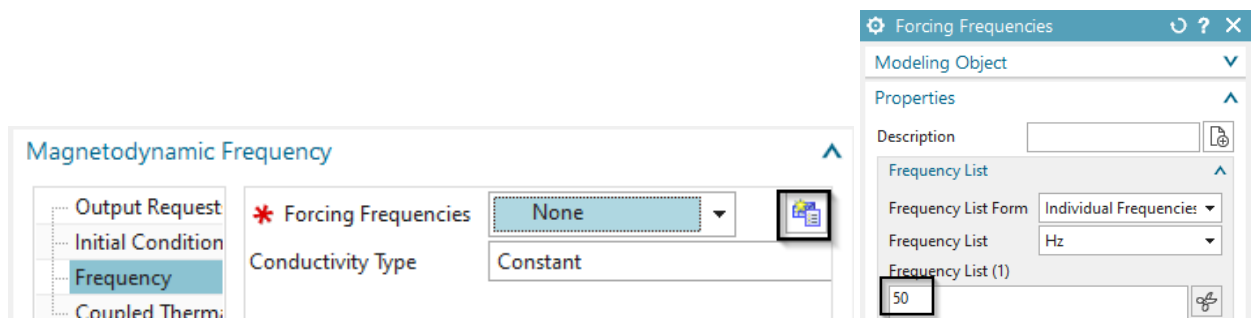
1. Open the part 'coil3d.prt',
  2. Start Simcenter Pre/Post, Create New FEM and Sim,
  3. Choose solver 'MAGNETICS' and Analysis Type '3D Electromagnetics'.
- Set Solution Type to 'Magnetodynamic Frequency'.



- In register 'Output Requests' under 'Table' activate the shown options

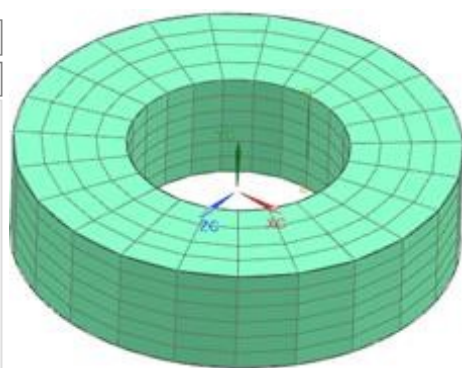
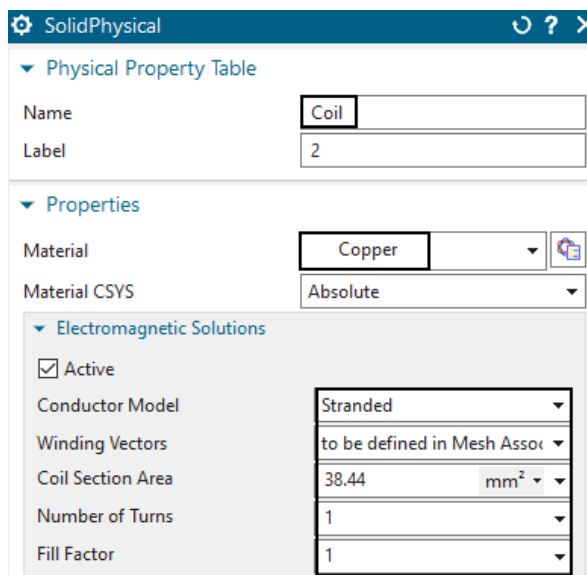


- In register 'Frequency Domain', create a modeling object and set the 'Forcing Frequency' to 50 Hz.

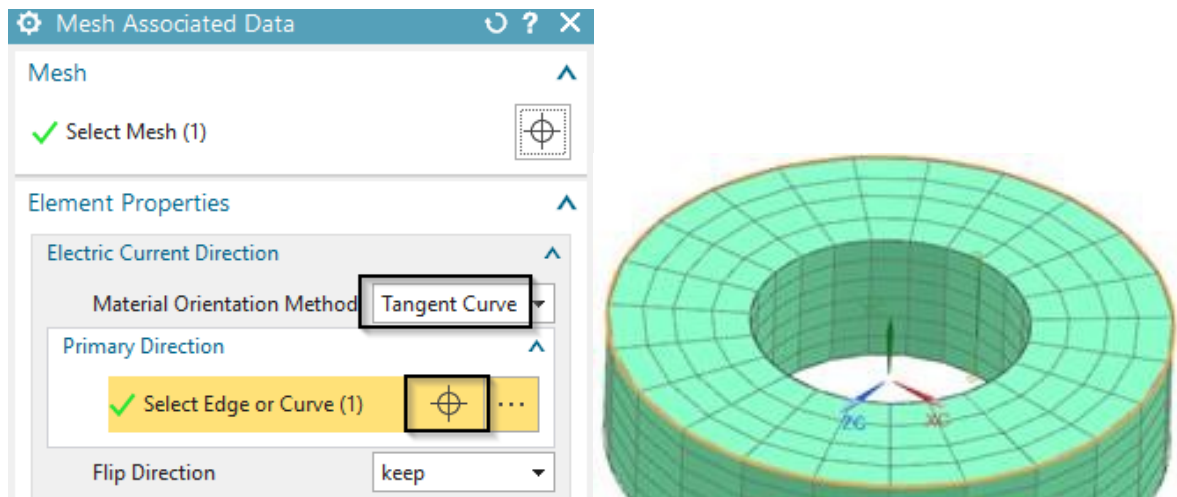


#### 4. Switch to the Fem-file.

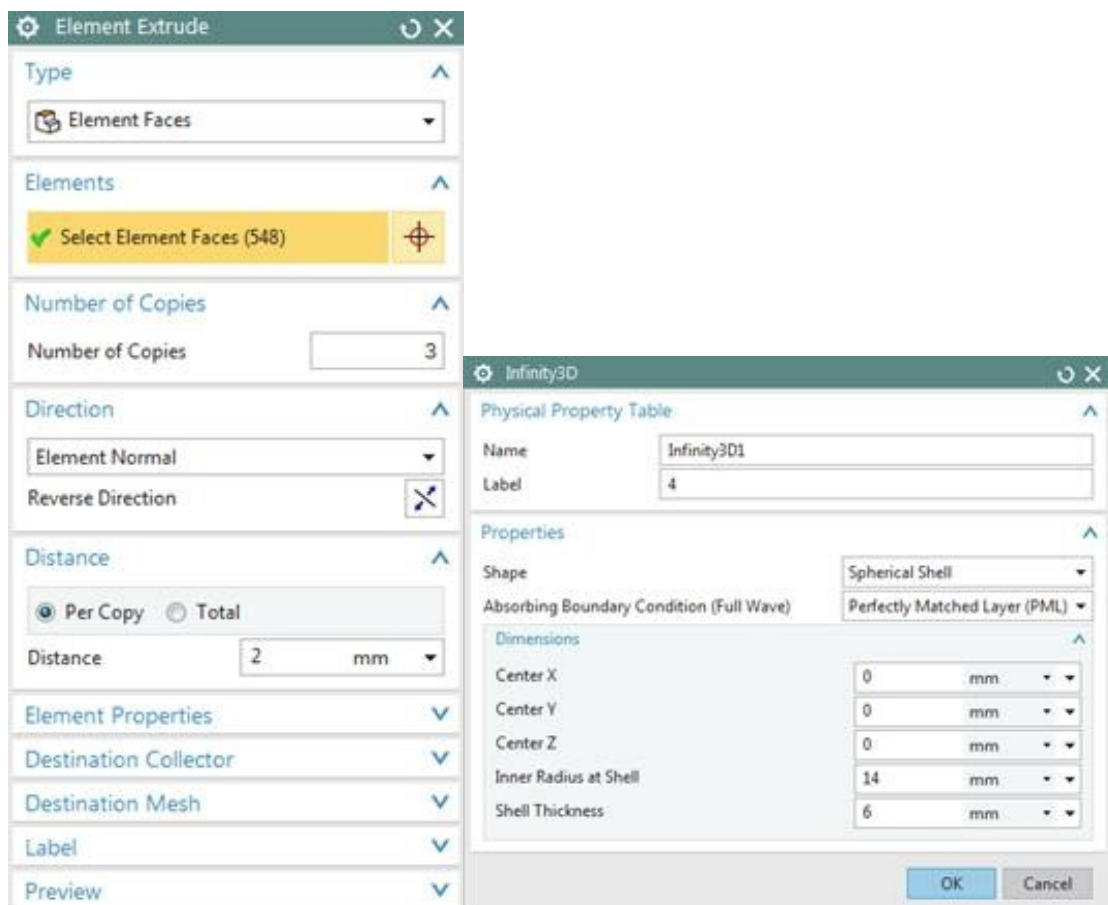
- Create Mesh-Mating Conditions.
- Hex-mesh the coil: Use the half of the suggested element-size ( $4.62/2$  mm). Assign material 'Copper' and the settings as shown in the next picture.



- Assign vectors to the coil mesh.



- Tet-mesh the air: Use the suggested element-size (4.92 mm). Hint: Activate the 'Pyramid Transition' option for correct transition between hex and tet elements.
- Assign material 'Air'.
- Use the function 'Extrude' from toolbar 'Nodes and Elements' to create 3 layers of infinity elements. Use a distance of 2 mm for each layer. Insert the settings to the Infinity physical as shown in the picture.

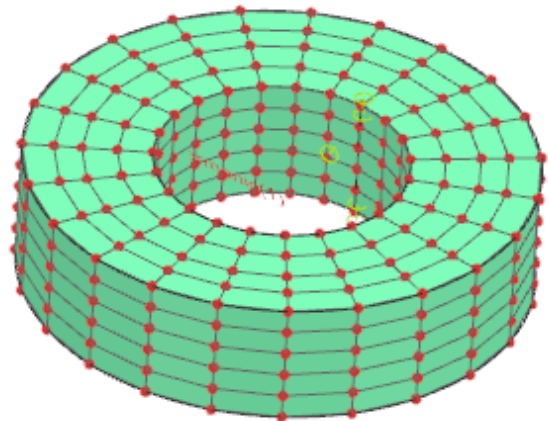
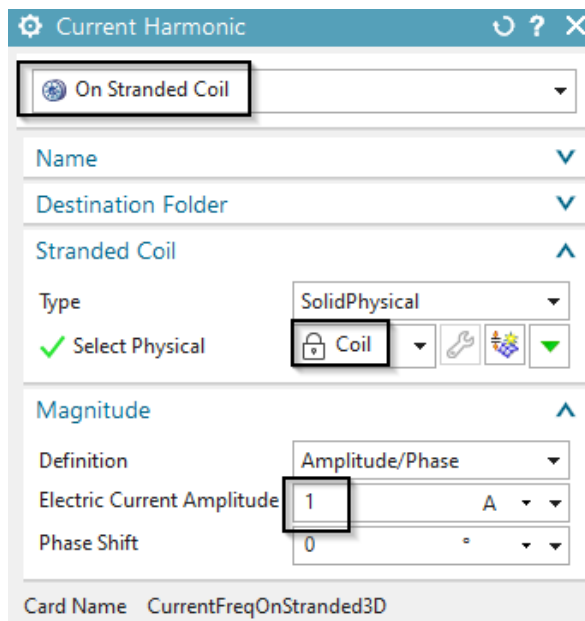


## 5. Switch to the Sim-file

- Create a constraint 'Zero Potential - Flux tangent' on the outside element-faces of

the sphere.

- Create a load of type 'Current Harmonic' on any of the faces of the coil. Assign 1 A and zero phase shift.



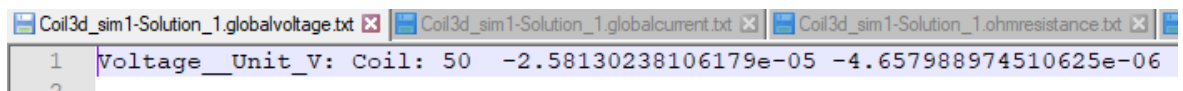
- Under Solution, select 'Edit Solver Parameters' and modify the field Result Graphs (afu) to 'Create, keep txt Files'
- Solve the solution

### 5.2.7 Post Processing

Postprocessing is done by some text files that contain the results.

#### 1. Voltage Result

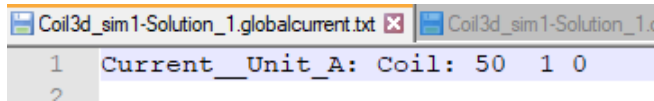
- Open \*.globalvoltage.txt .
- There are two values for each region. (the very first value is the forcing frequency) The first is the calculated 'Active Voltage' (Wirkspannung) and second is the 'Reactive Voltage' (Blindspannung). On region 'Coil' the result is
- 'Active Voltage' (Wirkspannung):  $-2.58\text{e-}5$  V
- 'Reactive Voltage' (Blindspannung):  $-4.66\text{e-}6$  V .



#### 2. Current Result

- Open \*.globalcurrent.txt.
- Again there are two results for each region. The first is the 'Active Current' (Wirkstrom) and the second is the 'Reactive Current' (Blindstrom). Because in our example we have a series circuit (Reihenschaltung) of inductivity and ohm-resistance the reactive current has to be zero. The sum of all active currents has to be zero, region

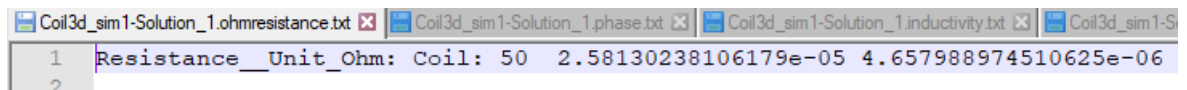
'Coil' has 1 Ampere.



1	Current__Unit_A: Coil: 50	1	0
2			

### 3. Ohm-Resistance Result

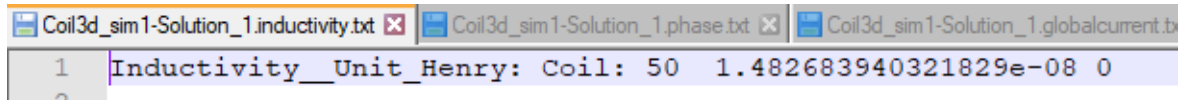
- Open \*.ohmresistance.txt.
- Again there are two results for each region. The first one is the 'Active Resistance' (Wirkwiderstand) and the second is the 'Reactive Resistance' (Blindwiderstand). The resistance is derived from the voltage and current results by  $R = U/I$ .
- This ohm resistance result is valid for a coil with one winding. For a coil with N windings the result values have to be multiplied by  $N^2$ .



1	Resistance__Unit_Ohm: Coil: 50	2.58130238106179e-05	4.657988974510625e-06
2			

### 4. Inductance Result

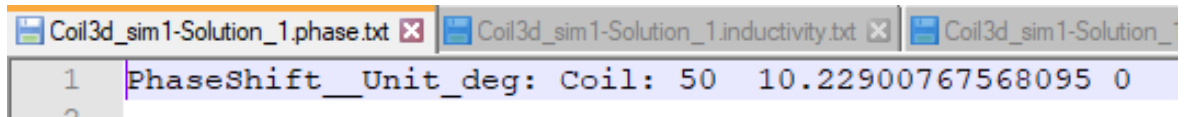
- Open \*.inductivity.txt.
- Again there are two results for each region, but in this case the second one does not have a meaning. The Inductance is derived from the voltage and current results by  $L = (U/I)/(2\pi \cdot f)$ .
- This result is valid for a coil with one winding. For a coil with N windings the result value has to be multiplied by  $N^2$ .



1	Inductivity__Unit_Henry: Coil: 50	1.482683940321829e-08	0
2			

### 5. Phase Shift Result

- Open \*.phase.txt.
- Again there are two results for each region, but in this case the second one does not have a meaning. The phase shift is derived from voltage results by  $\Phi = \arctan(U_{\text{ima}}/U_{\text{real}})$ .



1	PhaseShift__Unit_deg: Coil: 50	10.22900767568095	0
2			

#### 5.2.8 Result Comparison with Analytic

For approximate analytic calculation of the inductance  $L$  of an air-coil, we use the following formula (Source: G. Schenke, Bauelemente der Elektrotechnik 2008, S. 37.)

$$L \approx \mu_0 N^2 0.51 D_{\text{out}} = 1.589 \cdot 10^{-8} H$$

with  $D_{\text{out}} = 0.0248m$ : Diameter of coil,  $N = 1$  Number of turns,  $\mu_0 = 4\pi 10^{-7}$  Magnetic constant

The active ohm resistance  $R$  (Re part) can be calculated from the length  $l$  and section area  $A$  of the wire and the electric conductivity  $\sigma$  as

$$R = \frac{l}{\sigma A} = 2.62 \cdot 10^{-5} \Omega$$

with  $\sigma = 58 \cdot 10^6 S/m$ ,  $l = 0.058m$ ,  $A = 3.8438 \cdot 10^{-5} m^2$

The reactive ohm resistance  $XL$  (Im part) can be calculated from the inductance  $L$  and the frequency  $f$  from

$$XL = \omega L = 2\pi \cdot f \cdot L = 4.99 \cdot 10^{-6} \Omega$$

with  $f = 50Hz$ ,  $L = 1.589 \cdot 10^{-8} H$ .

The phase shift  $\varphi$  results from the two resistance values by

$$\varphi = \arctan \frac{Im}{Re} = 10.7deg$$

Finally, an overview of the calculated results in 2D and 3D and analytic is shown.

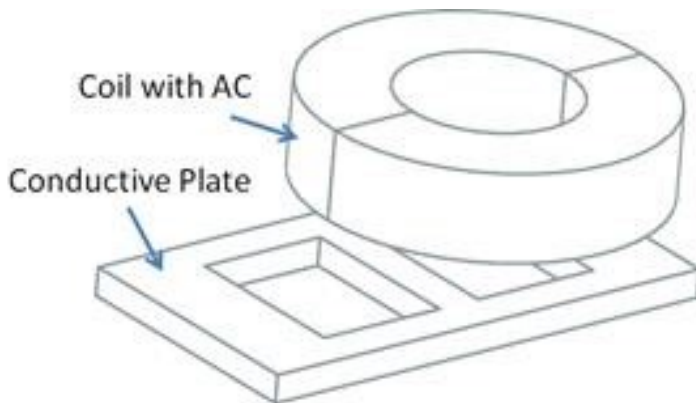
	Ohm Resistance (active)	Ohm Resistance (reactive)	Inductance	Phase Shift
Analytic	$2.62 \cdot 10^{-5} \Omega$	$4.99 \cdot 10^{-6} \Omega$	$1.589 \cdot 10^{-8} h$	10.7 deg
2D	$2.614 \cdot 10^{-5} \Omega$	$4.938 \cdot 10^{-6} \Omega$	$1.572 \cdot 10^{-8} h$	10.698 deg
3D	$2.581 \cdot 10^{-5} \Omega$	$4.658 \cdot 10^{-6} \Omega$	$1.483 \cdot 10^{-8} h$	10.23 deg

As another reference, the benchmark example 'TEAM 15' gives an inductance value for a similar coil with N=3790 windings as  $0.2218h$ . In our 2D example we calculated  $0.2258h$ . Thus, the deviation in 2D is 1.8% and in 3D 4%.

The exercise is done. Save your files and close them.

### 5.3 Conductive Plate under Coil (TEAM 3)

The problem 3 of the Team Benchmarks (Testing Electromagnetic Analysis Methods) is one of the examples for testing eddy current effects. A conductive plate with two holes is placed under a coil. The coil is driven by alternating current of 50 Hz and 1260 ampere turns. The goal is to analyse for the magnetic flux density along a line that goes slightly over the plate. The user should read the documents 'problem3.pdf' and 'Results\_EddyCurrentProblems.pdf' for more background understanding.







What you learn in this example:

- Perform 3D Magnetodynamic analysis in frequency domain,
- Display and calculate the Skin depth.

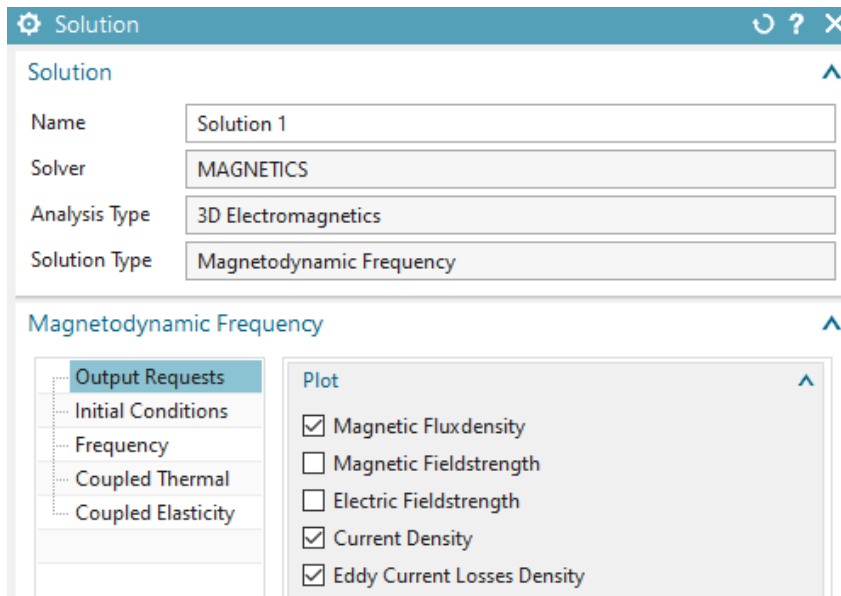
Estimated time for the example: 1h.

#### 5.3.1 Analysis of Complete Geometry

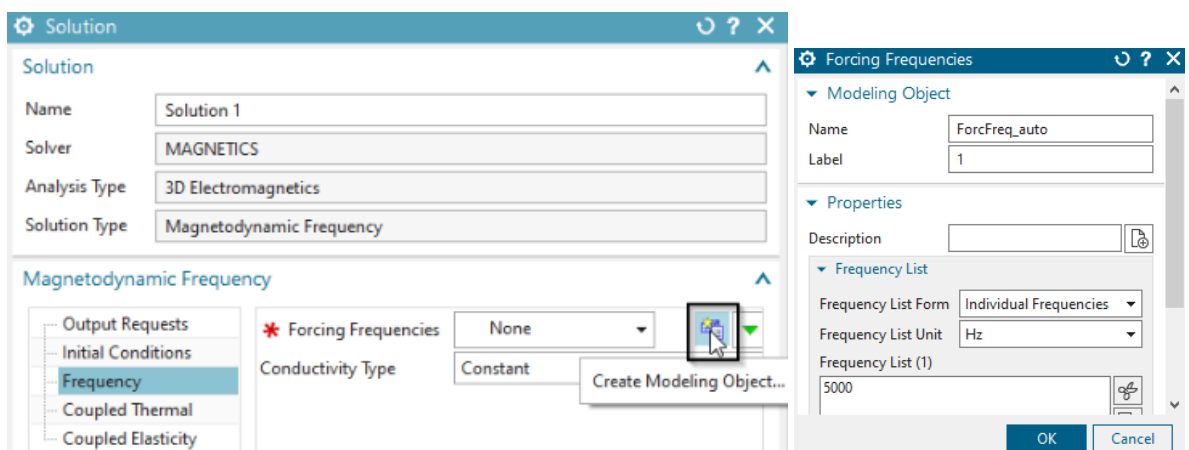
Follow these steps:

1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.3Team3.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program Simcenter  (or NX). Use Version 10 or higher.
4. In Simcenter, click Open  and navigate to folder 'start'. Select the file 'Team3\_full.prt' and click OK. (Maybe you must set the file filter to 'prt')
5. From toolbar Application click on 'Pre/Post' .
6. Click in toolbar 'Magnetics' on 'New FEM and Simulation (Non-Manifold)' .
  - Create a new Solution
  - Accept the Solver 'MAGNETICS' and choose the Analysis Type '3D Electromagnetics',
  - Choose Solution Type '3D-Magnetodynamic Frequency'

- In register 'Output Requests', 'Plot' set 'Current Density' and 'Eddy Current Losses Density' on to compute eddy current effects.



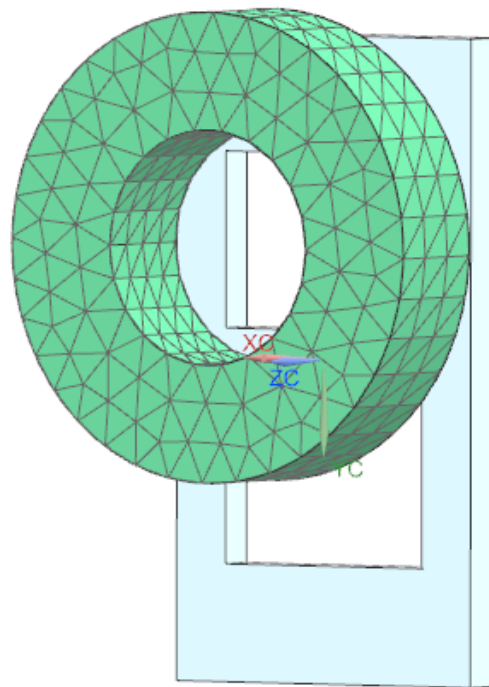
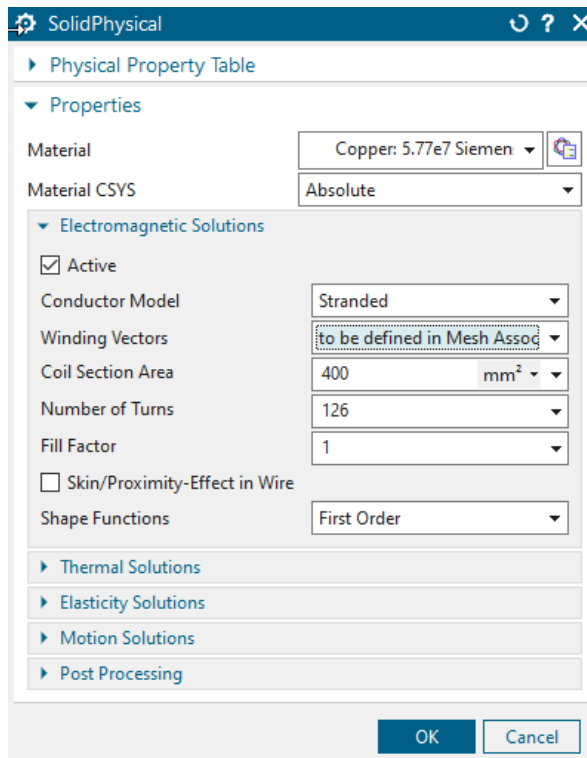
- In register 'Frequency Domain', click 'Create Modeling Object'. In the following window 'Forcing Frequencies', key in 5000 Hz into the list and click OK.



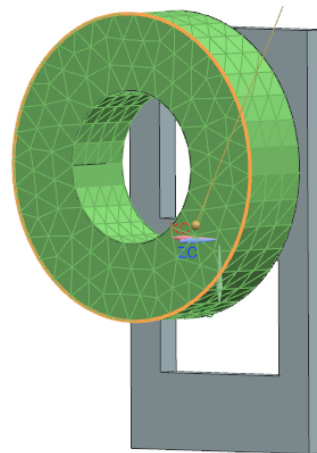
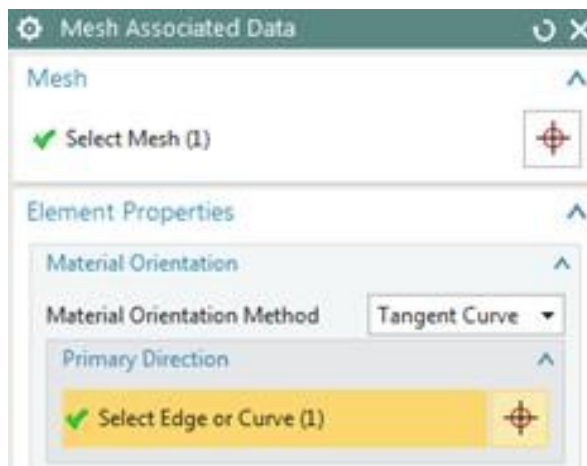
7. Set the displayed part to the Fem-file.

8. The Coil:

- Create a 3D tetra mesh on the coil body. Select the coil, key in the source element size (half of the suggested). Click OK.
- Edit the physical of the coil and assign the material Copper to it.
- Set the 'Conductor Model' option to 'Stranded'. The 'Number of Turns' to 126 and the 'Coil Section Area' to  $400\text{mm}^2$ .



- Edit the Mesh Associated Data for the coil mesh. Set the 'Material Orientation' to 'Tangent Curve' and select the circular edge to describe the winding direction.




#### 9. The Plate:

- Create a 3D tetra mesh on the plate. Use the system suggested value/4.
- Edit the physical and click 'Choose Material'.
- Click on 'Create Material' (Bottom right).
- Under 'View', switch to 'All Properties'. As name, type in 'PlateSample'.
- Under 'Electromagnetics' set 'Model' to 'Low Frequency'.
- For 'Electrical Conductivity', type in  $32760000 S/m$ . And for 'Relative Permeability', key in 1. Click OK several times until the window disappears.

The screenshot shows the 'Electromagnetic' properties panel. On the left, a list of property categories includes Mechanical, Strength, Durability, Formability, Thermal, **Electromagnetic**, Creep, Viscoelasticity, Viscoplasticity, Damage, and Other Physical Properties. The 'Electromagnetic' section is expanded, showing 'Model' set to 'Low Frequency'. Under 'Electrical Properties', 'Conductivity' is selected over 'Resistivity', and 'Linear' is selected over 'Nonlinear'. The 'Electrical Conductivity' is set to 32760000 S/m. A button 'Calculate Conductivity from %ACS' is visible. Under 'Magnetic Properties', 'Soft Magnet' is selected over 'Permanent Magnet', and 'Linear Permeability' is selected over 'Complex Permeability' and 'Nonlinear Permeability'. The 'Relative Permeability' is set to 1.

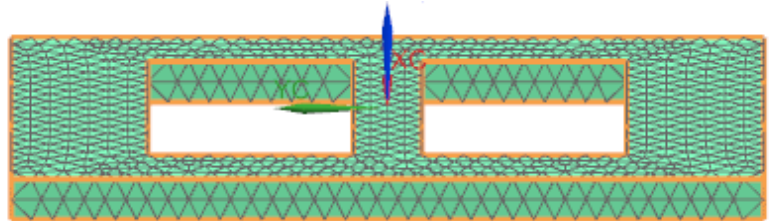
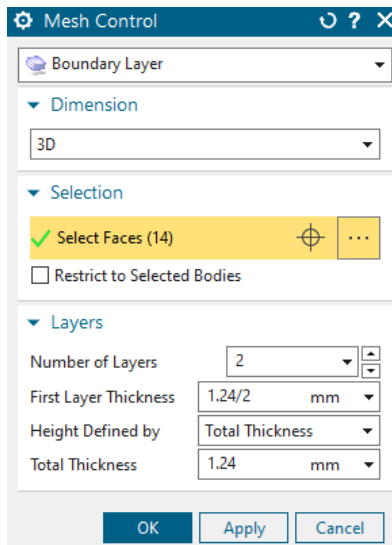
## 10. Mesh Control


- Click on 'Mesh Control' .
- Assign 'Boundary Layer'
- Select all plate faces (drag a window).
- Under 'Number of Layers', key in 1, 'First Layer Thickness', 1.24 and 'Total Thickness', 1.24. We do this to consider the skin depth in this simulation. Click OK.

Hint: Because of the skin effect, eddy currents will appear mainly at the border of the conductor. The thickness of that eddy current layer is called Skin Depth. For a accurate simulation this skin depth should be meshed with high quality elements. At least one layer of elements as we do in the following. The skin depth ( $\delta$ ) can be calculated by the formula:

$$\delta = \frac{1}{\sqrt{\pi \cdot f \cdot \mu_0 \cdot \mu_r \cdot \sigma}} = 1.24mm$$

with  $f = 5000Hz$ ,  $\mu_0 = 4\pi \cdot 10^{-7}$ ,  $\mu_r = 1$ ,  $\sigma = 32760000 \text{ S/m}$ .



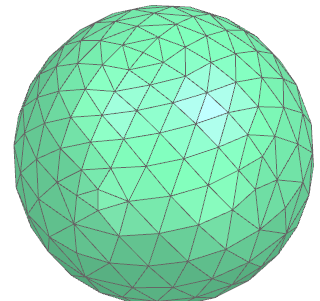
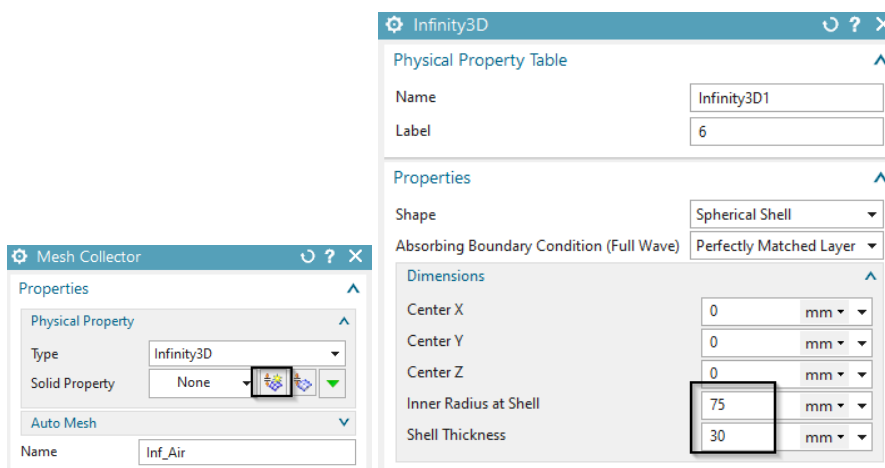
- Update the FEM .

#### 11. The Air

- Create a 3D tetrahedral mesh with the suggested size (24.1 mm) on the air volume. Use the existing mesh collector 'AIR (3d)'.

#### 12. The Air Infinity

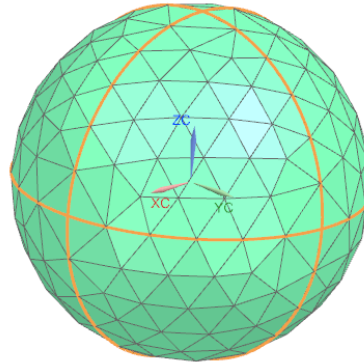
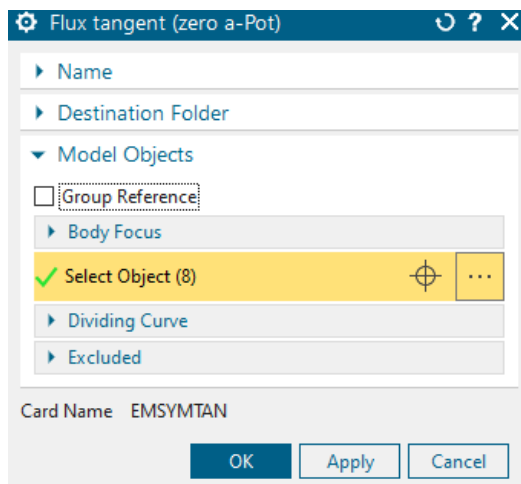
- This step is optionally for higher result accuracy but it can only be done if there is a geometry for the infinity air (not the case in the geometry file): Create a 3D tetrahedral mesh and select the body 'AirInf' (not available in current model but can be created in the prt file) and accept the suggested size. Activate 'Automatic Creation'. Click OK.
- Create for this mesh a physical of type 'Infinity3D' and
- Insert the settings into the physical as shown.



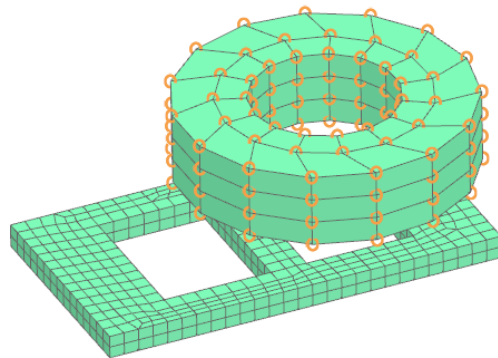
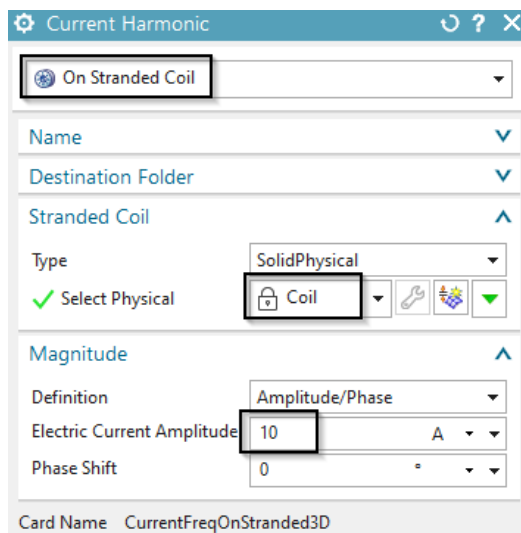
#### 13. Switch to the Sim-file

- Create a constraint of type 'Flux tangent (Zero-a-Pot)' on the outside element faces of the Air Infinity sphere. Set the selection filter to 'Tangent Faces' and select a face.

Click OK.



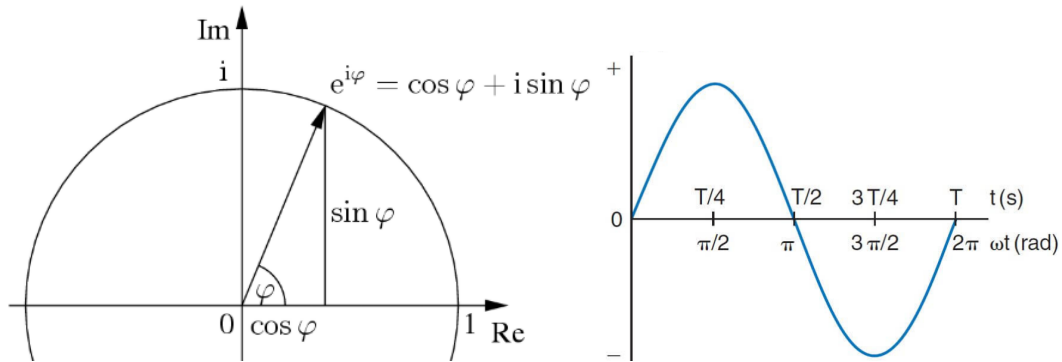
- Create a load 'Current Harmonic' with type 'On Stranded Coil' on the coil. Assign 10 A.



- Solve the solution. The solution time is about 1 minute.

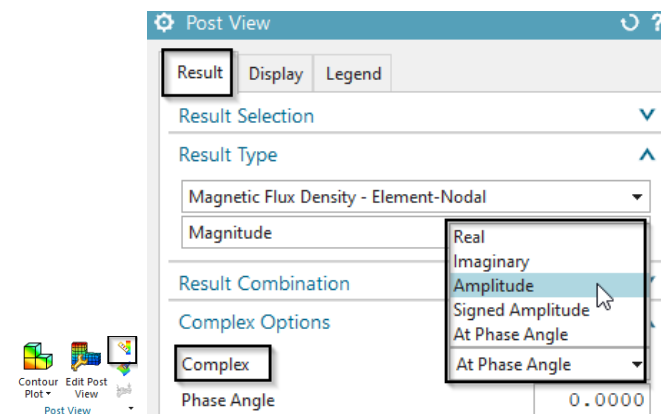
### 5.3.2 Post Processing Complex Results

In all frequency domain solutions results are complex, that means they have a real (Re) and an imaginary (Im) part. One can imagine the corresponding time domain solution would alter between this real and imaginary part. From real and imaginary part other quantities like amplitude and phase angle can be extracted by complex number math operations.

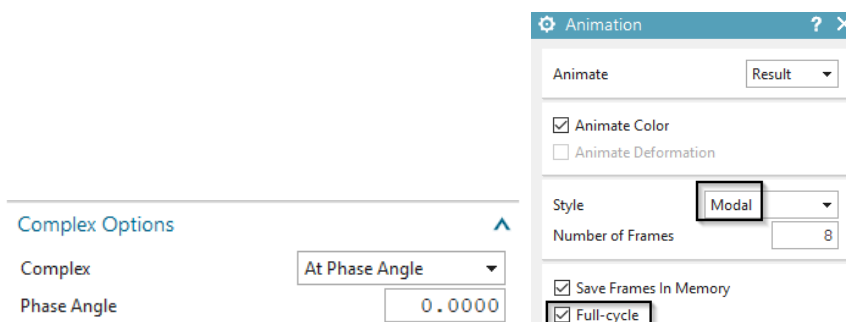


Hints for viewing frequency results in post processing:

- All frequency results are available in complex format what means the NX postprocessor can handle these comfortably. The 'Complex Options' can be set by first clicking 'Result' and then modifying the 'Complex'. That allows to switch between 'Real', 'Imaginary', 'Amplitude' and 'At Phase Angle'.



- The complex results of frequency solutions can be shown as time results. To do so, an animation over the 'Phase Angle' must be done. This will show, step by step, how the real part changes to the imaginary part, thus, the corresponding time solution. Proceed as follows:
- Set the option 'Complex' to 'At Phase Angle'.
- Click 'Animation' and set the 'Style' to 'Modal'. Activate 'Full Circle'. Then click 'Play'. The animation will show the corresponding time result.

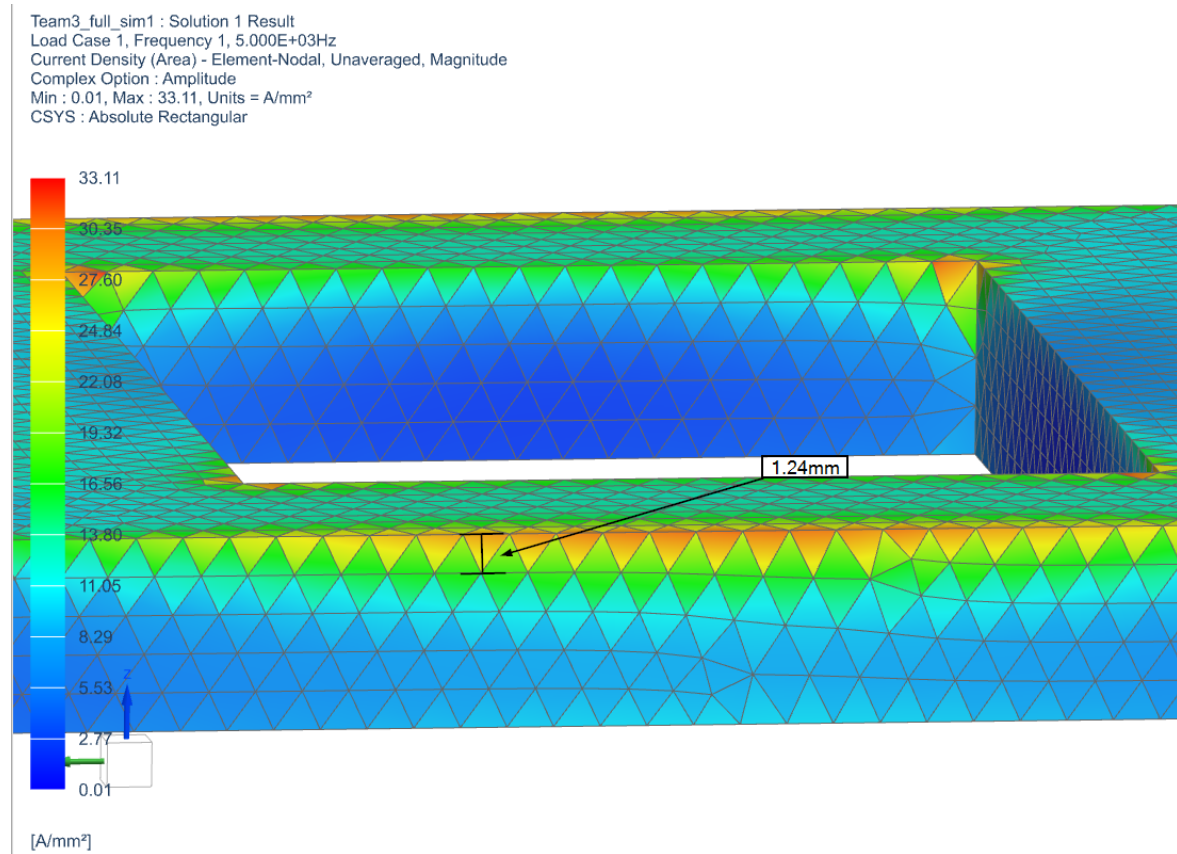


1. In the 'Post Processing Navigator' double click on the 'Current Density' result.

2. Hide the 3D elements Coil, Air, AirInf and all 2D elements.

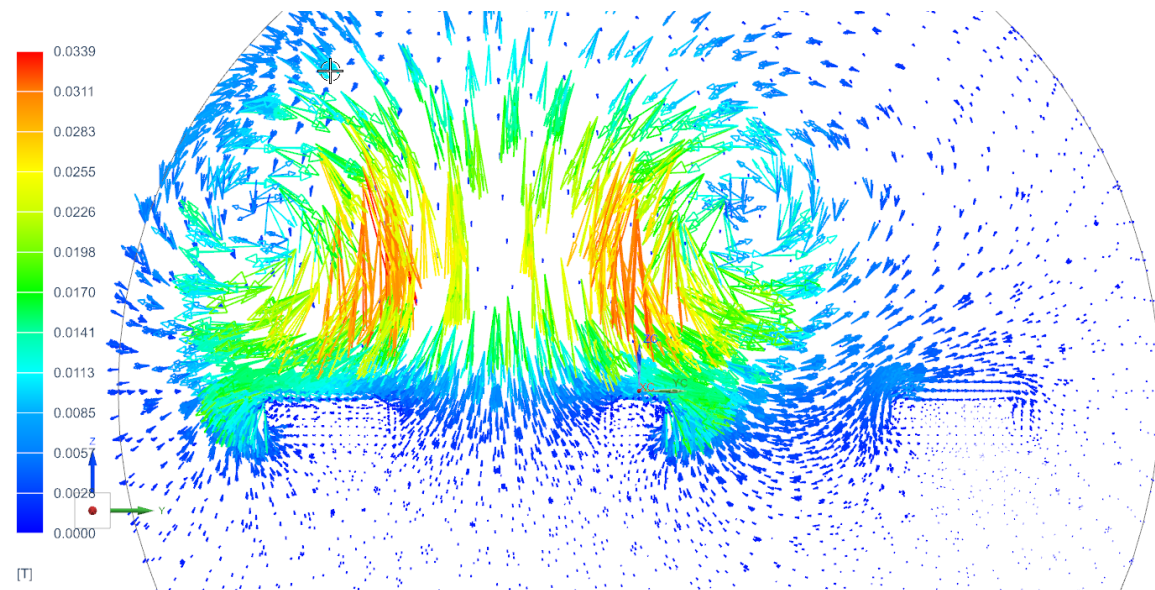
3. Current Density Result:

The picture shows the area where the eddy currents appear. It corresponds to the prior calculated skin depth.



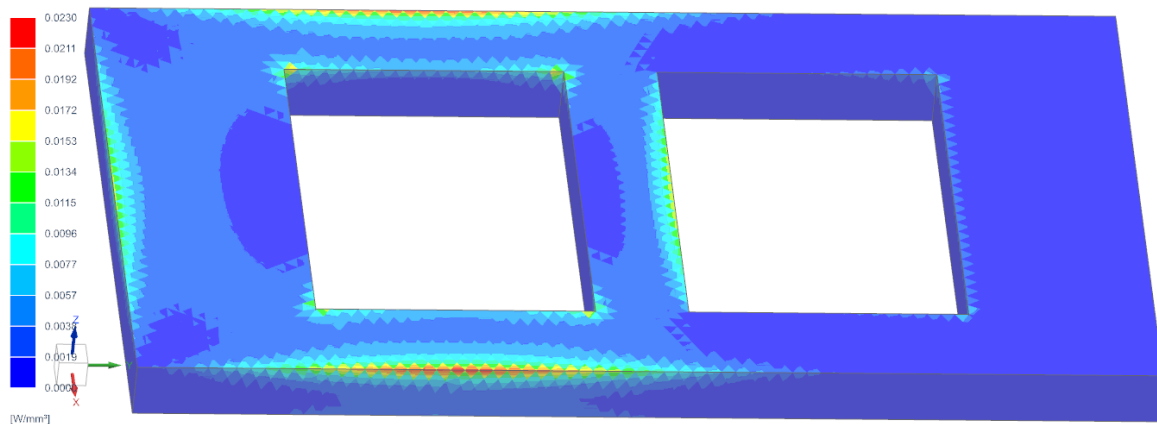
The picture shows the area where the eddy currents appear.

4. Flux density plot with arrows and the 'Complex Option' is set to 'At Phase Angle'. Visible is the magnetic field, created by the coil and influenced by the plate.



#### 5. EddyCurrentLossesDensity:

This result is responsible for the heating of the plate.

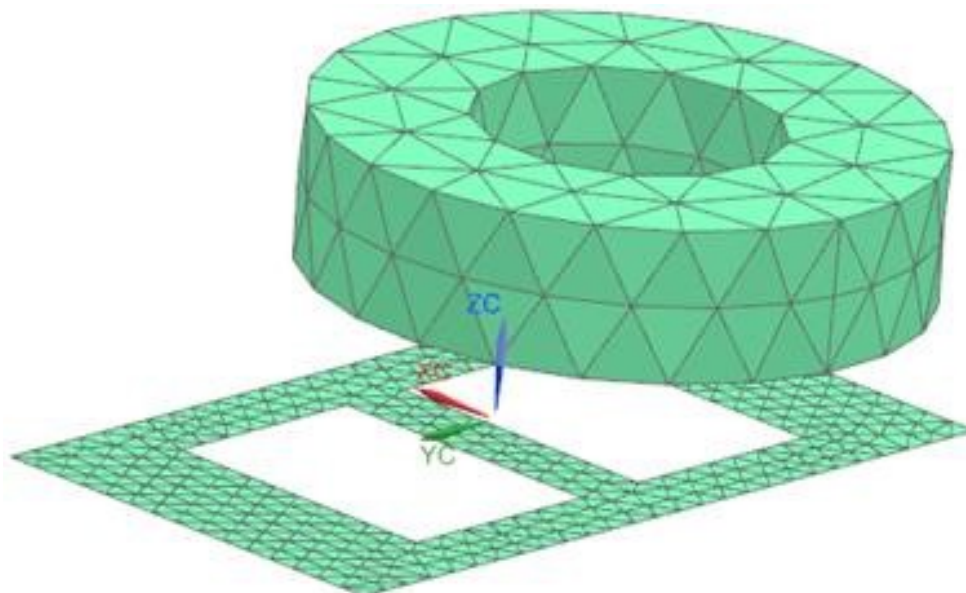


This result is responsible for the heating of the plate.

#### 5.3.3 Modeling the Plate by 2D Elements

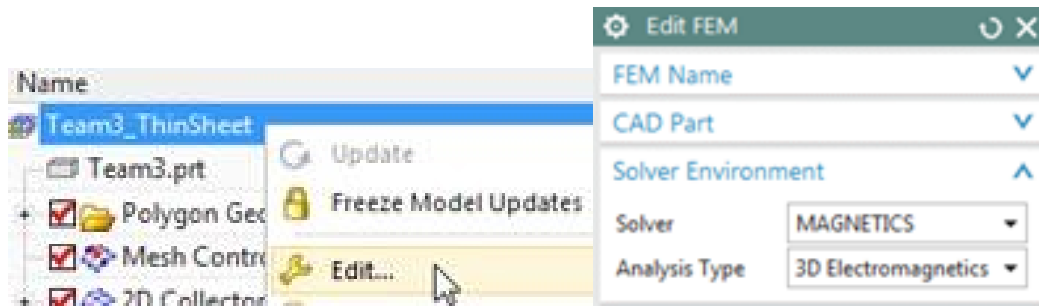
In this second part of the exercise we will demonstrate how conducting sheets can be used. Therefore, the aluminum plate will be modeled by 2D elements which take into account thickness and magnetic/electric material properties. This assumption is valid and of good accuracy if thickness is small. The model is already set up, so in this tutorial we simply walk through the steps and check the new features.

1. Open the part 'Team3\_sim2.sim' from the complete folder and change to the FEM file. The part already contains a coarse mesh, physical properties and boundary conditions for this exercise.

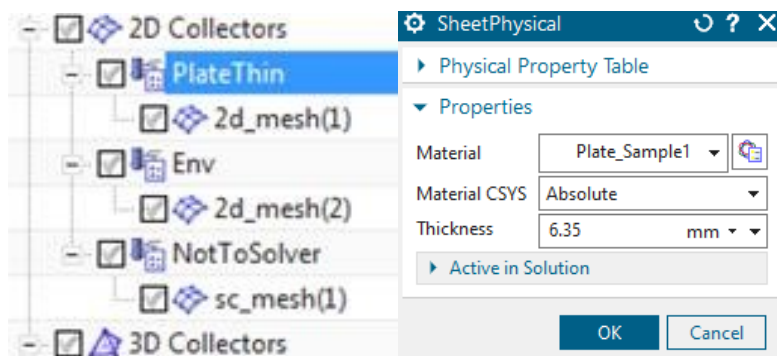


2. Set the Fem file to the displayed part.

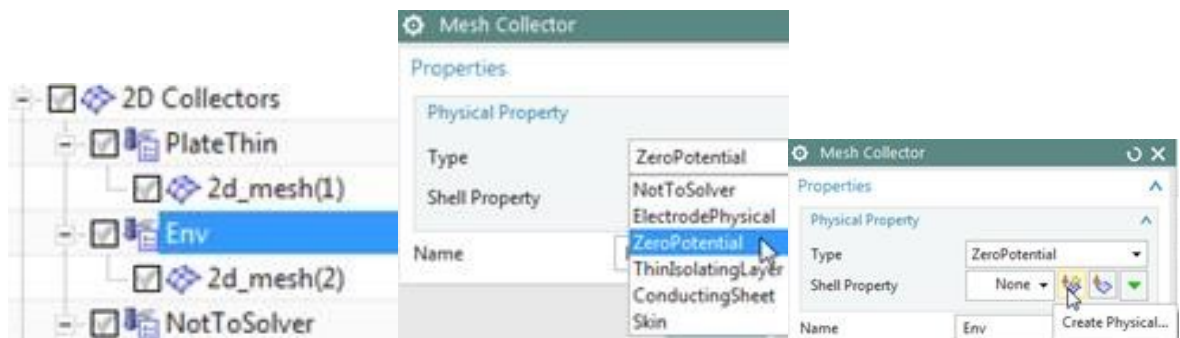
- Edit the FEM file and verify the solver is set to MAGNETICS and the analysis type is '3D Electromagnetics'.



- Notice the plate is meshed by 2D elements. The mesh resides in a collector named 'PlateThin'. Check the physical of the plate. It is of type 'SheetPhysical' and materials and thickness properties are defined as shown in the picture below:



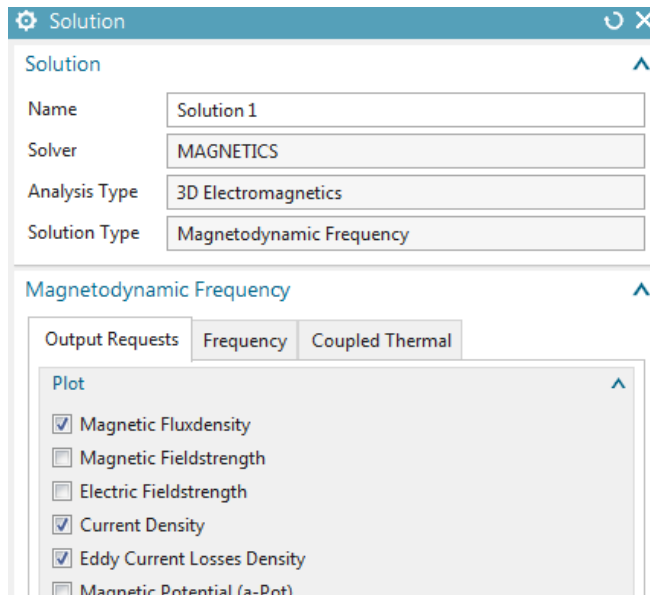
- Notice that in this example we used a 'Solid from Shell Mesh' for the air. Therefore, a 'Surface Coat Mesh' on all 3D parts exists (collector 'NotToSolver') and a tri mesh on the air boundary (collector 'Env').
- Check the 2D mesh collector 'Env'. It contains a physical of type 'ZeroPotential' for the boundary. This physical has the same effect as a constraint of type 'Flux tangent (zero a-Pot)' in the Sim file. You can use this alternatively. Its advantage is a little more performance when writing out the solver input file.



- Check the physical for the coil mesh. All properties are set in the same way as in the first part of this example (126 turns,  $400\text{mm}^2$  section area). Also the definition of the winding directions is already there.
- Check the FluidPhysical for the air mesh.

### 3. Change to the Sim file.

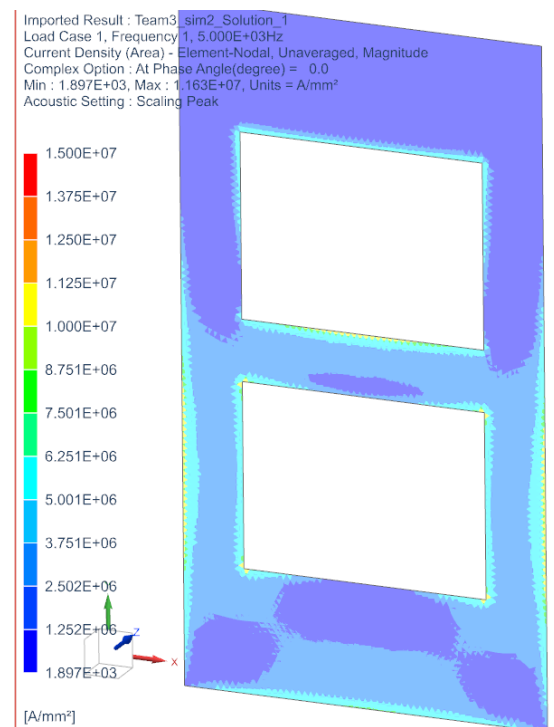
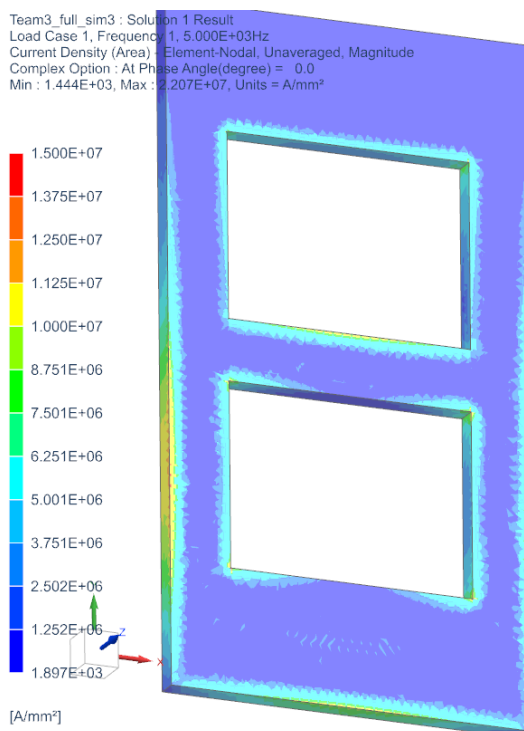
- The solver is Magnetics, Analysis type is 3D and Solution Type is 'Magnetodynamic Frequency'. All settings are set in the same way as in the first part of this example.



- The same current (10 A) is applied on the coil.
- There is NO 'Flux Tangent (zero a-Pot)' condition in the SIM file, because there is already a ZeroPotential physical that has the same function.

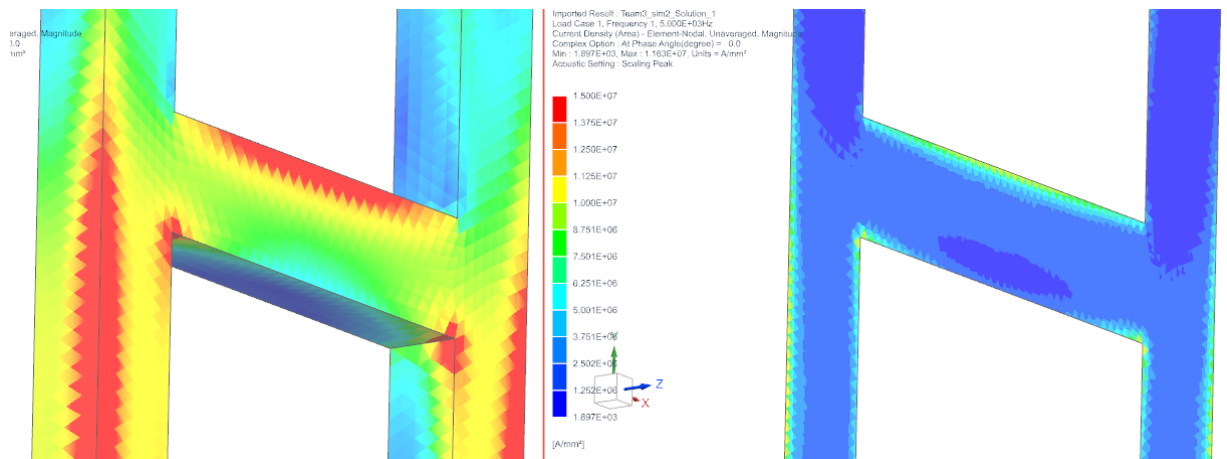
#### 4. Solve and post process the results.

- Check Current Density in comparison of the 3D plate with the 2D sheet: See next picture (left 3D plate, right 2D sheet):



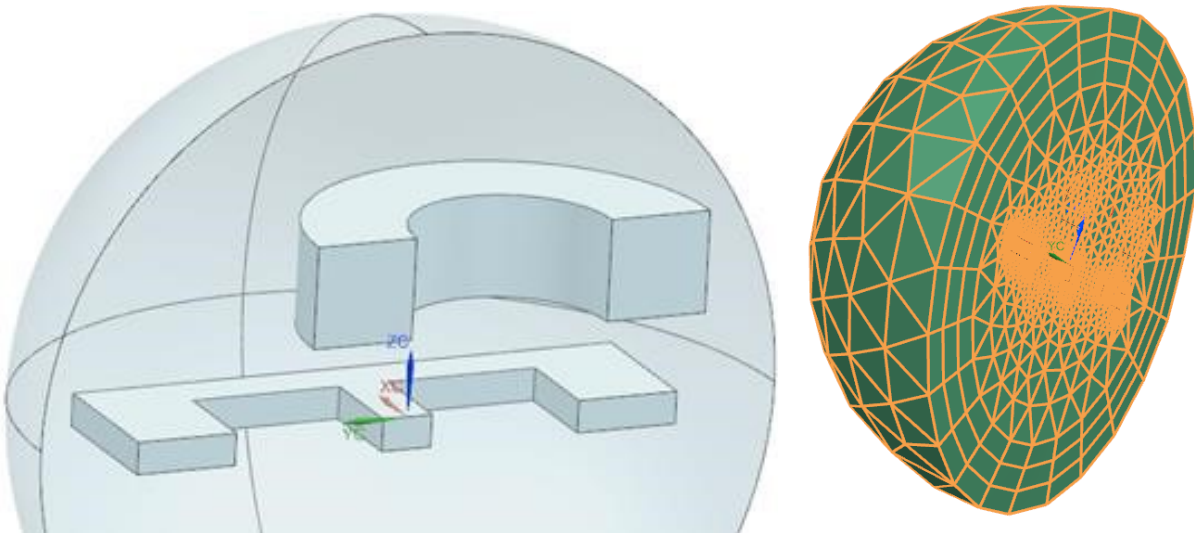
- The overall agreement is ok, but notice there are deviations between the 3D and the thin sheet results that result from edge effects. The 3D results are more precise. The skin effect is more detailed using the 3D plate model. Deviations become smaller with

thinner plates.



### 5.3.4 Using Symmetry

In case the model and also the magnetic field are symmetric, the simulation can be simplified. We use a half model and apply a symmetry constraint on the symmetry plane.



The condition 'Flux tangent (zero a-Pot)' on the symmetry plane acts as a symmetry condition. Therefore, that condition has to be defined on all outside faces: The outside sphere faces and the symmetry plane.

## 5.4 Shielding in Time Dependent Magnetic Field (TEAM 1a)

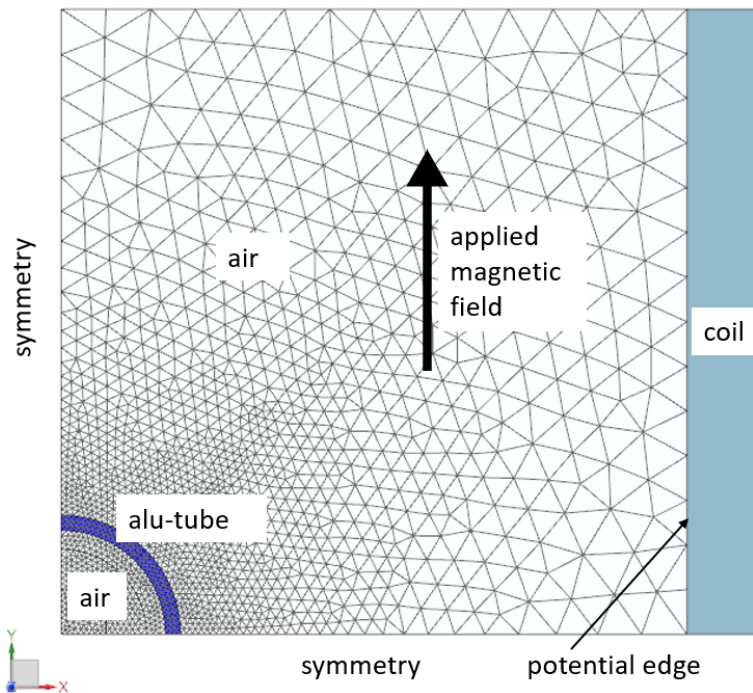
In this example an aluminum tube (depicted blue in the figure) will be considered that is loaded by a time dependent magnetic field in vertical direction. We will compute the time dependent induced current (eddy current) within the tube. Such eddy current losses can then be used to calculate temperatures on the tube (not shown in this tutorial). Reference results are available from the TEAM benchmark. Additionally, the reader may also read the attached document for the TEAM 1a task to find further information.

According to the TEAM 1a description, the applied magnetic field strength in the y direction decays exponentially with time as

$$B_y = B_0 e^{\frac{-t}{\tau}} \quad (1)$$

where  $B_0 = 0.1T$  and  $\tau = 0.0397$ .

For this example we will apply the demanded magnetic field in two different ways: first, we will apply said field via a difference of a magnetic potential on the two opposing boundaries; and second, we will apply an electric current on the coil face. In both cases the result shall be very similar. In order to start from a steady state solution we will first analyse for statics and then restart from this solution.





Estimated time for this example: 0.5 h

Exercises are:

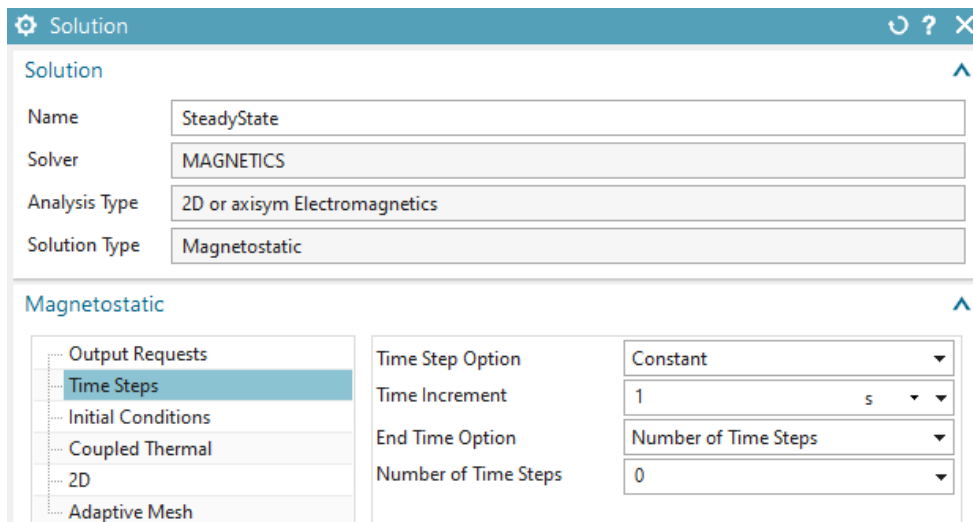
1. Analysis of Eddy Currents, conductive and hysteresis Losses,
2. Symmetry constraints,
3. Applying given potentials as math function,
4. Using a steady state solution as initial condition.

### 5.4.1 Solution with Potential Condition

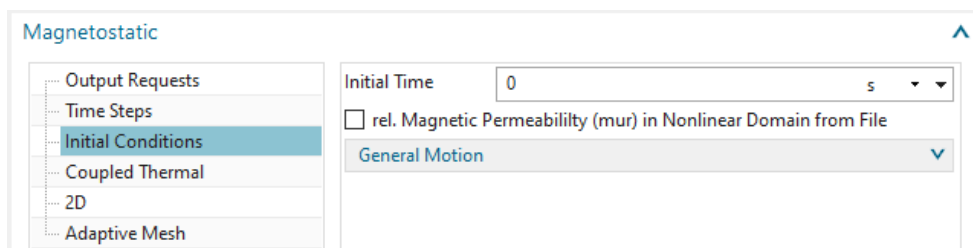
1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.1Team1a.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program Simcenter  (or NX). Use Version 10 or higher.
4. In Simcenter, click Open  and navigate to folder 'start'. Select the file 'Team1a.prt' and click OK. (Maybe you must set the file filter to 'prt')
5. Start application Pre/Post and create a 'New Fem and Sim' File.
6. Set off the creation of an Idealized Part.
7. Choose the Solver 'MAGNETICS', '2D or axisym Electromagnetics'.
8. Create a Solution of type 'Magnetostatic'.

Hint: This will be the steady state solution that we will later use to restart from.

- Name the solution 'SteadyState'.
- In register 'Time Steps', accept the defaults 'Number of Time Steps' as 0 as well as 'Time Increment' as 1.

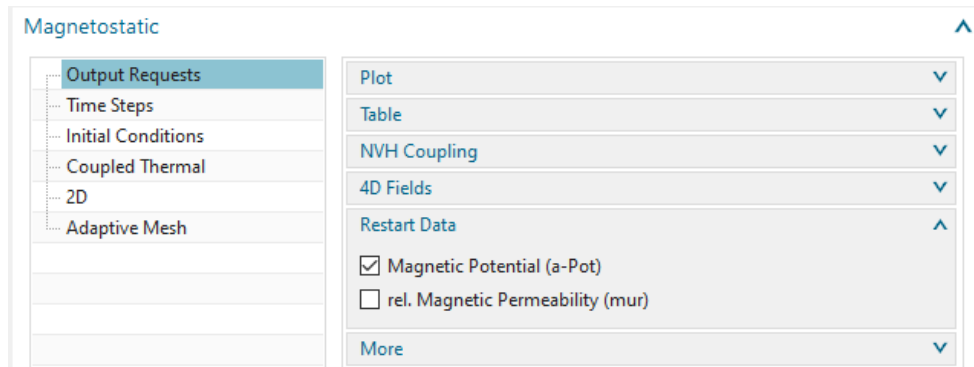


- In register 'Initial Conditions', accept the defaults, i.e. 'Initial Time' to 0 s.  
Hint: A second solution will restart from this first solution. Therefore, we want the second solution to start at time 0; meaning that we have to use zero time for this first solution.

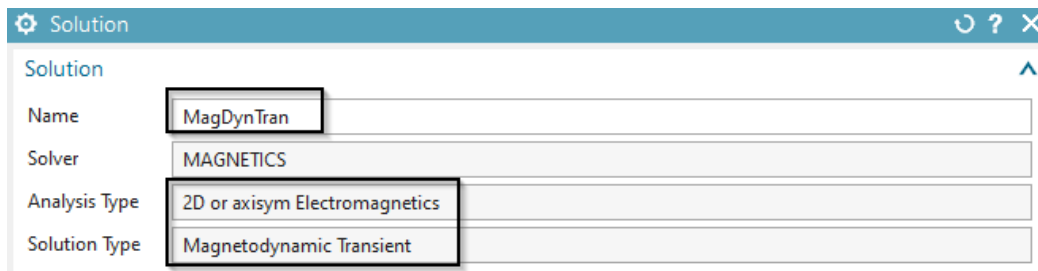


- Accept all other default settings.
- Edit the output requests of this solution. In box 'Restart Data', activate 'Magnetic Potential (a-Pot)'.

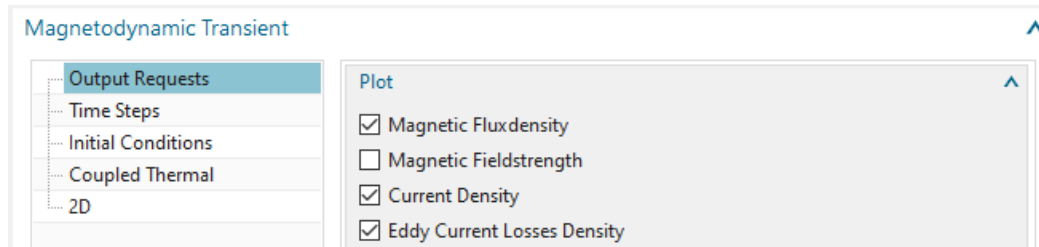
Hint: This setting is necessary to use this solution later to restart from.



9. Create a second solution of type 'Magnetodynamic Transient'. Name it 'MagDynTran'.  
Hint: This will be the main solution



- Activate the following Output Requests in register 'Plot':



- 'Current Density': To compute the induced eddy currents and display them as a contour plot.
- 'Eddy Current Losses Density': To compute the losses  $P_c$  that result from electric resistance and eddy currents  $I$  corresponding to  $P_c = RI^2$  with  $R$  as ohm resistance. This approach for losses analysis is valid typically in massive conductors where hysteresis effects are neglectable.
- And in register Table:

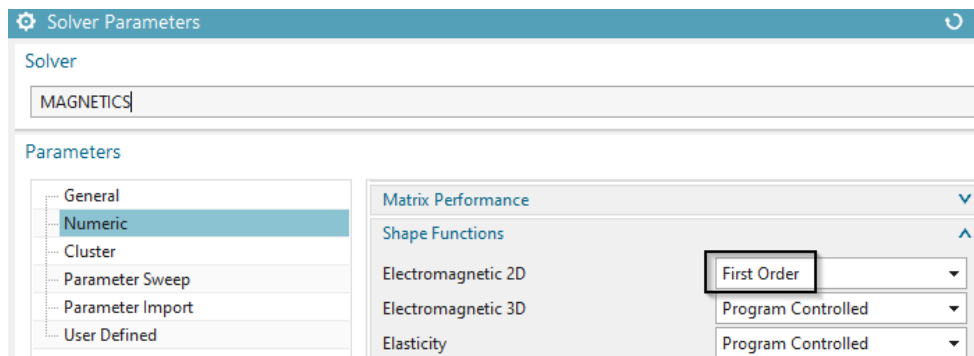
<input checked="" type="checkbox"/>	Electrode Current
<input type="checkbox"/>	Electrode Power
<input type="checkbox"/>	Circuit Voltage
<input type="checkbox"/>	Circuit Current
<input type="checkbox"/>	Circuit Power
<input checked="" type="checkbox"/>	Eddy Current Losses

- 'Eddy Current Losses': To compute the losses  $P_c = RI^2$  integrated over geometry
- Set the 'Time Steps' settings as in the next picture

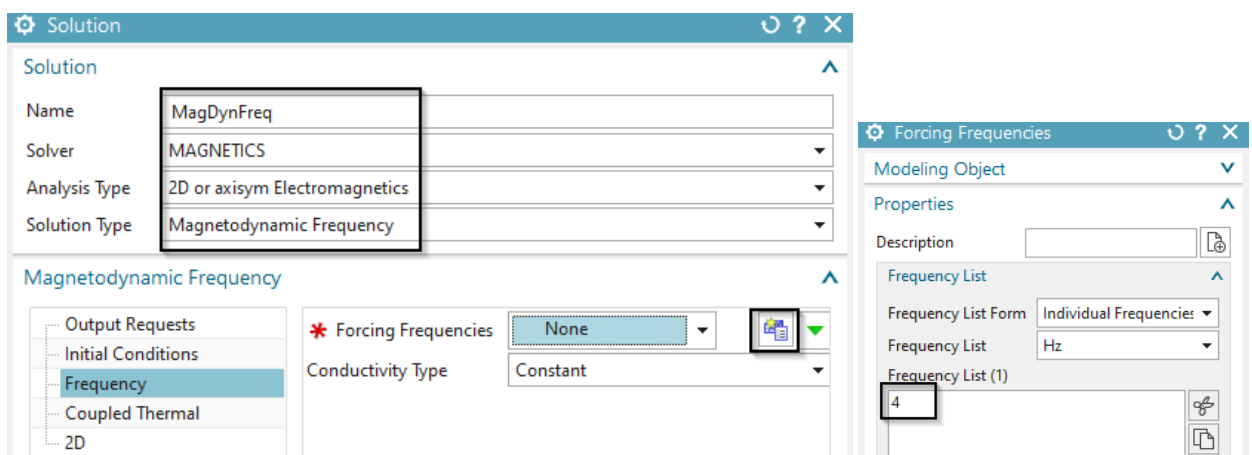
- In register 'Initial Conditions': Set the option 'Magnetic/Temperature' to 'Restart from other Solution'. Key in the name of the already created solution into the field 'Restart Solution Name'. Also activate 'Output' to enable the system to write the results for the initial condition to the result file. This is for checking only.

- Accept the defaults in register '2D'. Ok.

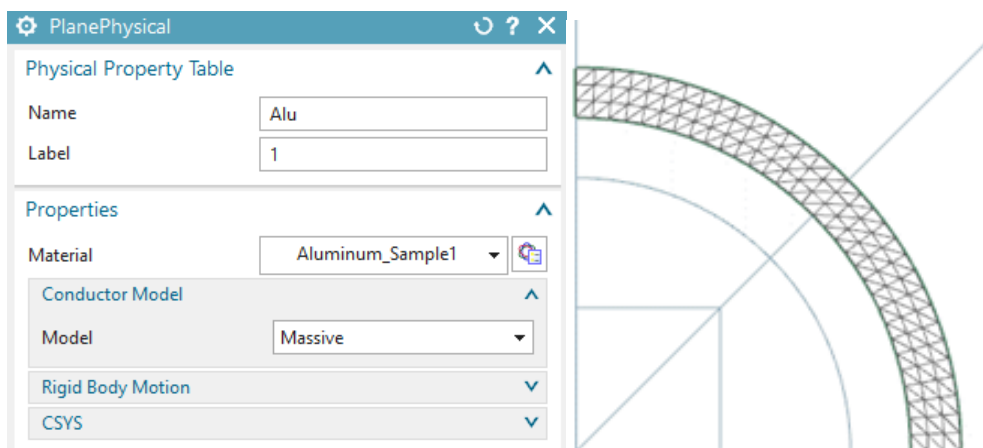
- To avoid a sharp step of eddy currents that would appear at the beginning: In solver parameters under 'Numeric', set the option 'Shape Functions, Electromagnetic' to 'First Order'.



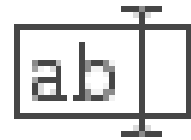
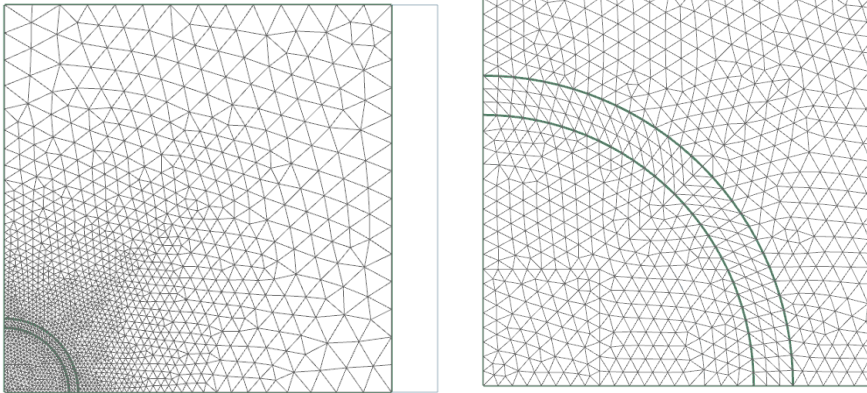
10. Optionally step: This problem can also be solved in frequency domain. The excitation has a frequency of about 4 Hz. So, if desired, create another 2D solution of type 'Magnetodynamic Frequency'. Name the solution 'MagDynFreq' and set the forcing frequency to 4 Hz. Constraints and loads can be made very similar to the transient one for this solution.



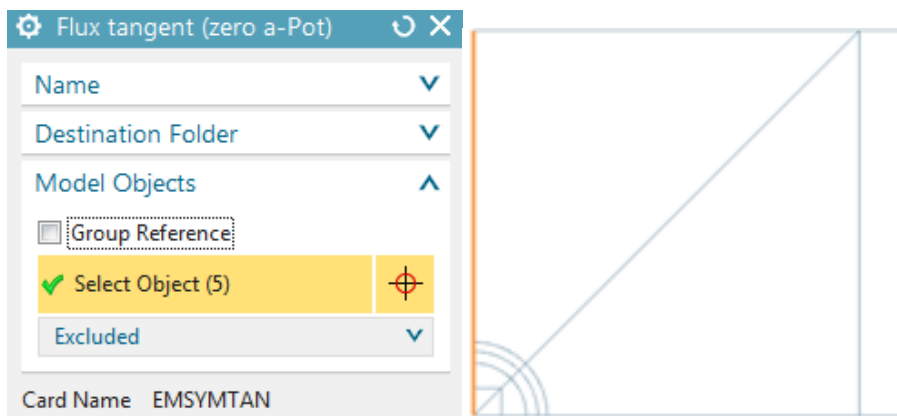
11. Switch to the Fem file.
12. Create a tri mesh on the faces of the aluminum tube. Use the suggested mesh size value divided by 2. The system automatically creates a 'PlanePhysical' which you can modify now.
13. Assign material 'Aluminum\_Sample1' from the library 'Magnetics\_Materials.xml'. Name the Physical and mesh collector 'Alu'.



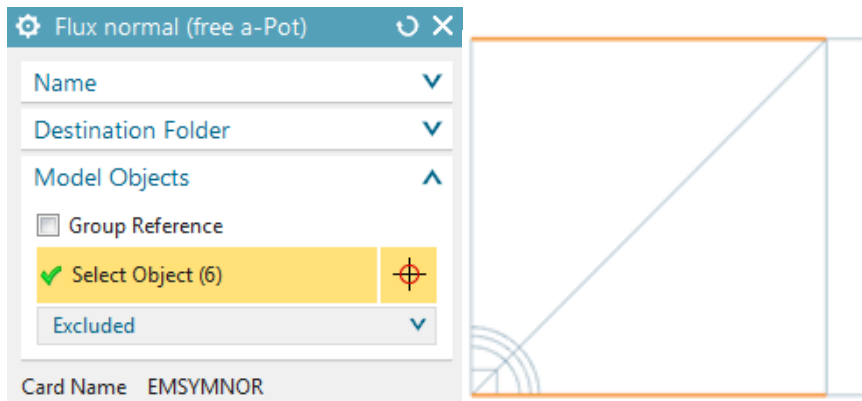
14. Create a tri mesh on the other faces (see picture, use suggested element size  $/2$ ) for the air. Assign a physical of type 'FluidPhysical' and 'Air' as material.  
Hint: Do NOT mesh the coil face. This will be used later. For this part of the tutorial the coil face stays unmeshed.



15. For easier post processing, click on the button 'Rename Meshes ...' from the Magnetics toolbar. That utility renames all meshes and therefore is a useful feature especially with larger models and many meshes.
16. Switch to the Sim file.
17. Create a first constraint of type 'Flux tangent (zero a-Pot)' for the symmetry at plane  $x=0$ .

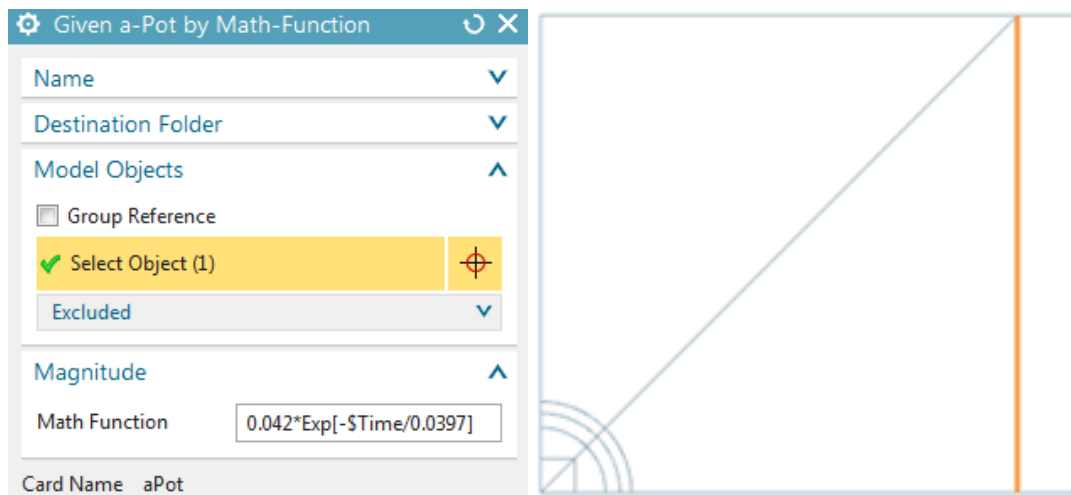


18. At plane  $Y=0$  (and at the top edge) we need normal flux lines. So create a constraint of type 'Flux normal (free a-Pot)' on these 6 edges.  
Background: These two conditions define the symmetric situation. Notice that the normal condition could also be left, because normal flux lines is the default behaviour of this magnetic vectorpotential formulation at boundaries.

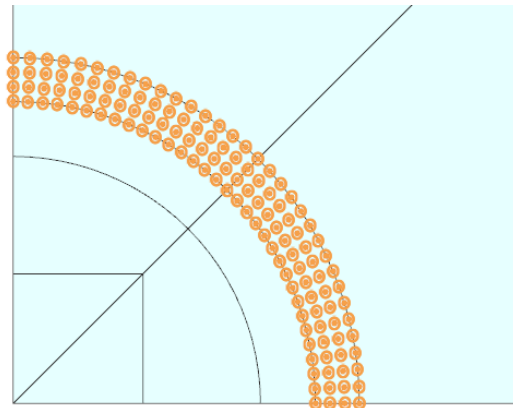
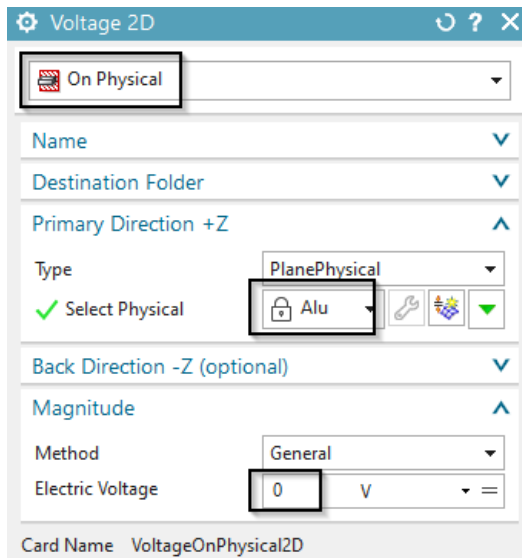


19. Define a time dependent flux density in the air.

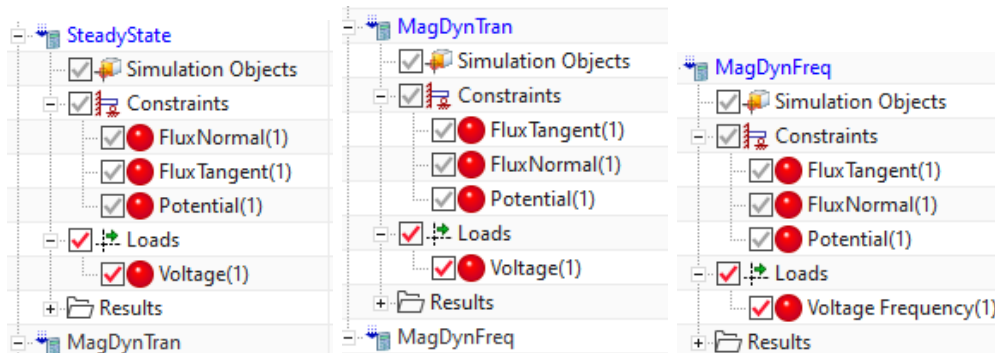
- Background: The task-description of TEAM 1a demands an exponential varying magnetic flux density with time. We create this flux density by using a given potential on the boundary edge. We use the trick that any magnetic potential difference between two positions results in magnetic flux through the area between the two positions. This constraint leads to an equally distributed flux density in a homogeneous area. Only the aluminum-tube will disturb the homogeneous flux density. And this disturbing effect shall be analysed.
- Create a constraint of type 'Given a-Pot by Math-Function' on the right edge. Key in the displayed formula into the field 'Math Function'. Click OK.



- Hints: The factor 0.042 results from the requirement that at the beginning of the time period a constant flux density of 0.1 T shall exist. This value was computed by a pre analysis without aluminum-tube. The syntax for this formula and further possibilities are described in the Getdp-documentation.
20. Create a zero voltage load with type 'On Physical' on the alu tube. This is necessary to fix the z direction degree of freedom for the electric current in this conductor. Such a condition must always exist on electric conductors in dynamic simulations. In case the optional frequency solution is there, this voltage load must be created for this solution additionally.

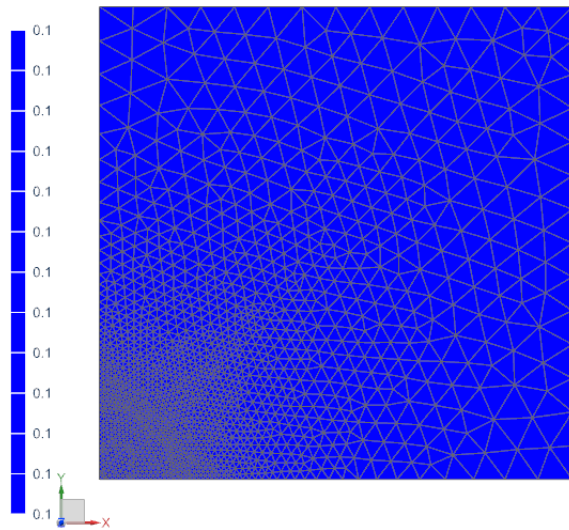


21. Put all constraints and the load into all solutions. The solutions, two or three - depending on the optional frequency solution - in the navigator should look like in the picture.

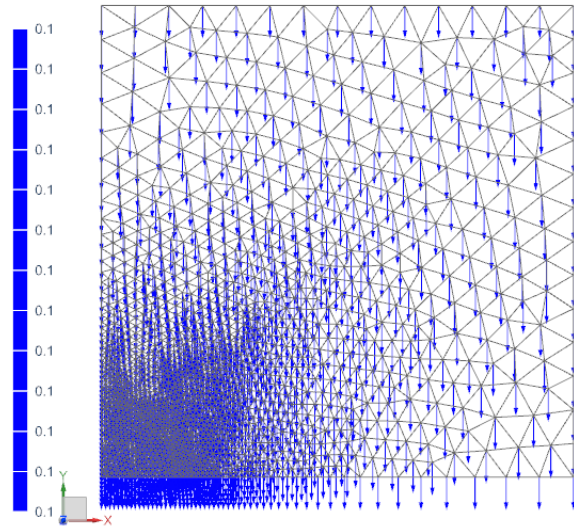


22. First solve the 'SteadyState' solution. Then solve solution 'MagDynTran' that restarts from the previous one.
23. Check the SteadyState result:
24. Check that at the beginning of the time period flux density is exactly 0.1 Tesla in the whole area. The directions are all vertically.

Team1a\_sim1 : SteadyState Result  
Load Case 1, Increment 1, 0s  
Magnetic Flux Density - Element-Nodal, Unaveraged, Magnitude  
Min : 0.1, Max : 0.1, Units = T



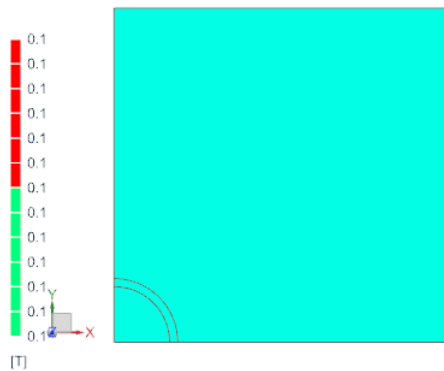
Team1a\_sim1 : SteadyState Result  
Load Case 1, Increment 1, 0s  
Magnetic Flux Density - Element-Nodal, Unaveraged, Magnitude  
Min : 0.1, Max : 0.1, Units = T



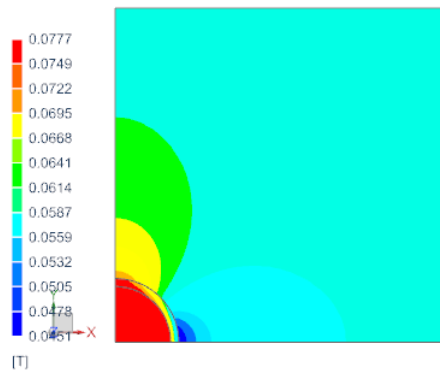
25. Then check the transient result:

26. Check how magnetic flux density varies with time. The following pictures show magnetic flux density at the beginning, in the middle and at the end of the time period.

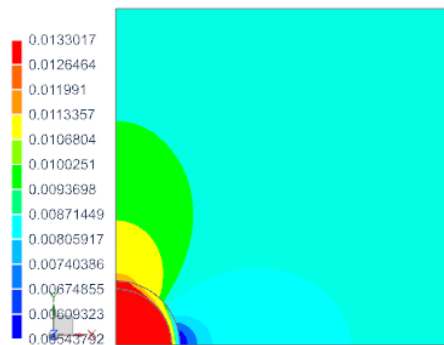
Team1a\_sim1 : MagDynTran Result  
Load Case 1, Increment 1, 0s  
Magnetic Flux Density - Element-Nodal, Averaged, Magnitude  
Min : 0.1, Max : 0.1, Units = T



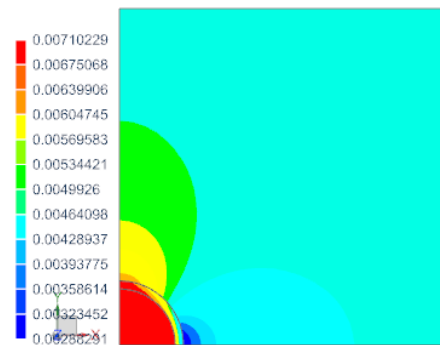
Team1a\_sim1 : MagDynTran Result  
Load Case 1, Increment 5, 0.02s  
Magnetic Flux Density - Element-Nodal, Averaged, Magnitude  
Min : 0.0451, Max : 0.0777, Units = T



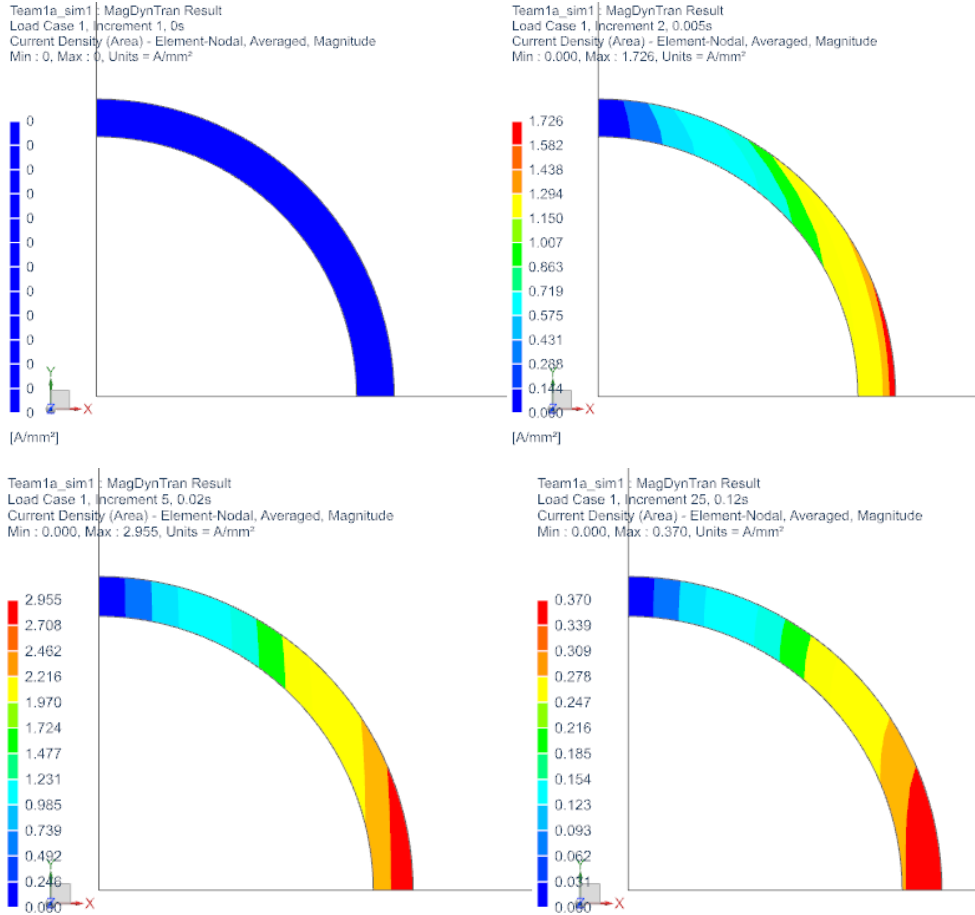
Team1a\_sim1 : MagDynTran Result  
Load Case 1, Increment 20, 0.095s  
Magnetic Flux Density - Element-Nodal, Averaged, Magnitude  
Min : 0.00543792, Max : 0.0133017, Units = T



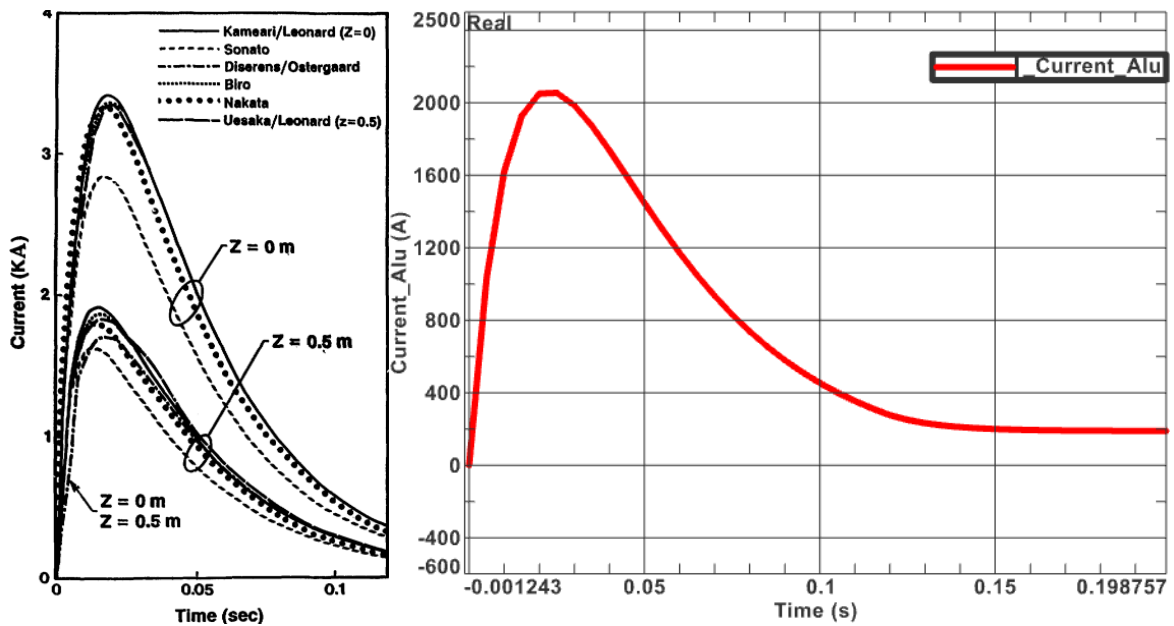
Team1a\_sim1 : MagDynTran Result  
Load Case 1, Increment 25, 0.12s  
Magnetic Flux Density - Element-Nodal, Averaged, Magnitude  
Min : 0.00288291, Max : 0.00710229, Units = T



27. Check the induced current density in the aluminum-tube at increments 1, 2, 5 and 25.



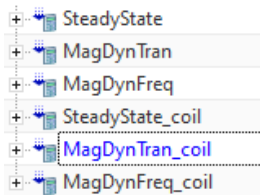
28. In the following picture, we see two different graphs which plot the current in the Alu over time. The graphs in the left picture result from several benchmark teams using different numerical codes, whereas the right one shows the simulated solution in our code NX-Magnetics. Since the left approaches are all centered around our simulation, we can say that our simulation has a derivation of about 10%.



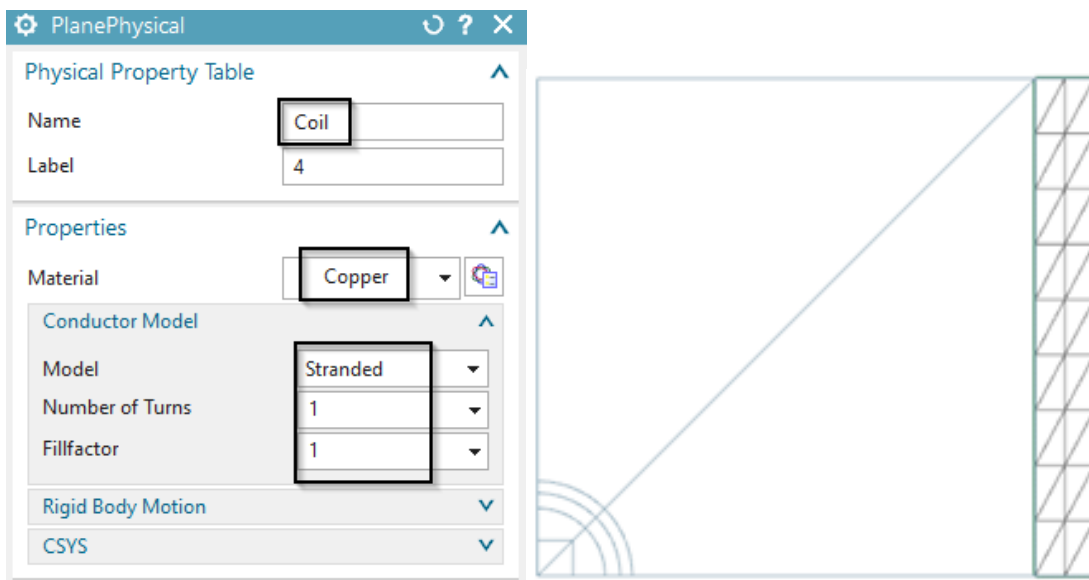
### 5.4.2 Solution with Electric Current Load


As an alternative to the first part of the example we want to realize the flux density condition now by giving a time dependent current on the right face. The resulting magnetic field leads to the same flux density.

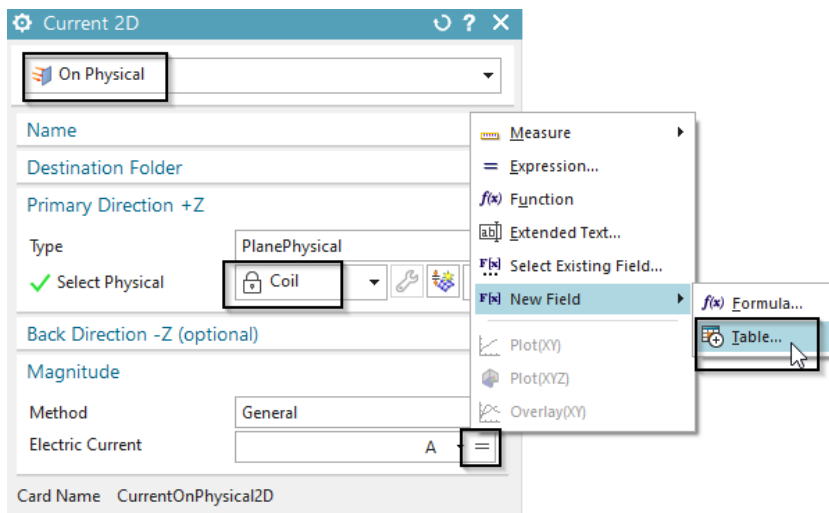
1. Open the previously created Sim file.
2. Clone all solutions (2 or 3, in case the frequency solution is also there). Rename the new ones with extension '\_coil'.



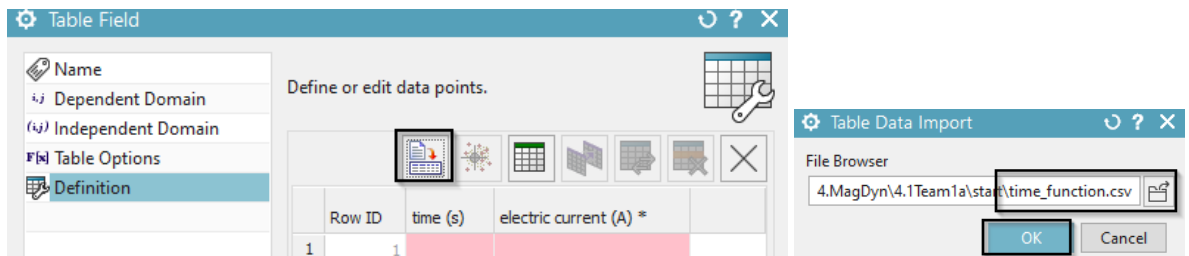
3. in the transient one, change the 'Initial Condition - Restart Solution Name' to the new one: 'SteadyState\_coil'.
4. Change to the Fem part and mesh the face right in the picture. Assign material copper and stranded properties with 1 turn. Using this stranded method you will get a homogeneous distribution of the current over the face.



5. Change to the Sim file and
6. In all new solutions (\_coil), remove the previously used potential constraint.
7. Create a load of type 'Current 2D', with type 'On Physical'.
  - Accept the default method 'General'.
  - Select the newly created physical 'Coil'.
  - Click the 'equal' button  and select the table function as shown in next picture.



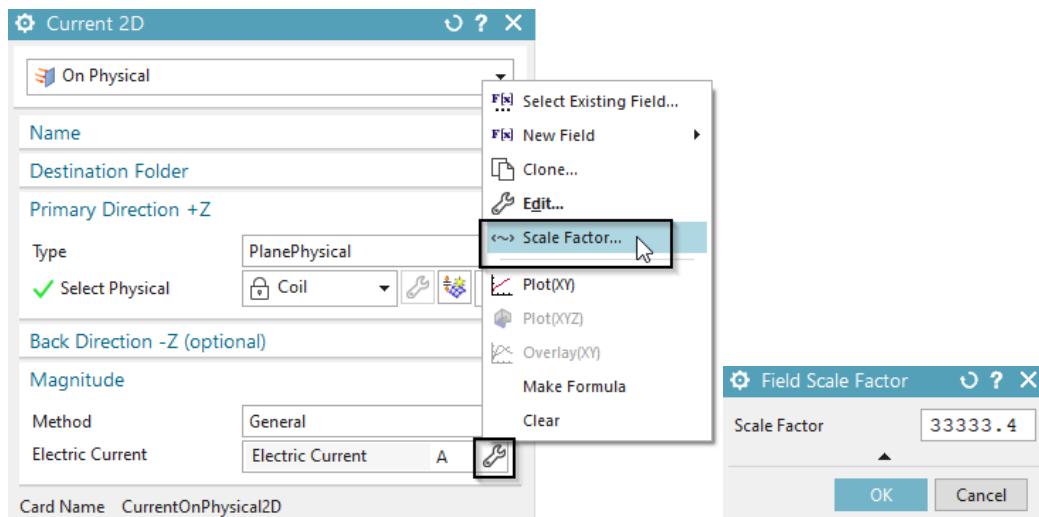
- In following dialogue 'Table Field' click 4 times 'Next' to accept some defaults. Then, click 'Import from File' and select the file 'time\_function.csv' from the 'start' folder, that already contains data points for the exponential function. OK.



- All the points are transferred into the dialogue. Click OK.

	Row ID	time (s)	electric current (A) *
1	1	0	1
2	2	0.005	0.8816637
3	3	0.01	0.7773309
4	4	0.015	0.6853444
5	5	0.02	0.6042433
6	6	0.025	0.5327394
7	7	0.03	0.469697
8	8	0.035	0.4141148
9	9	0.04	0.36511

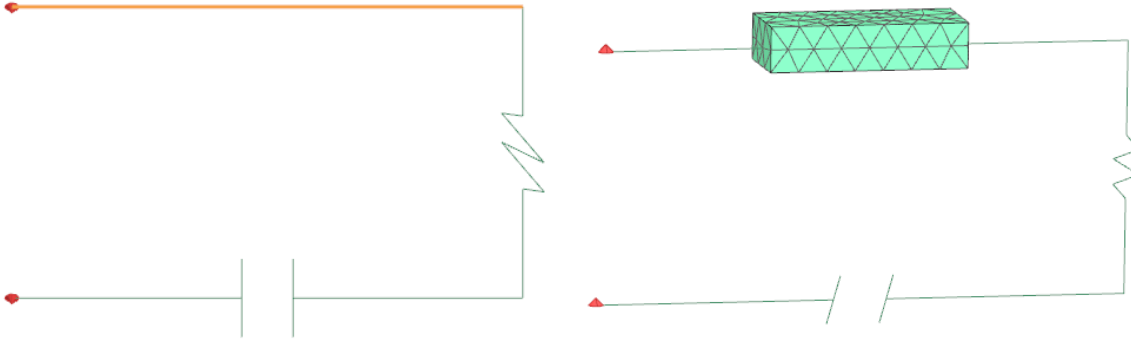
- Back in the prior dialogue, insert a scale factor of 33333.4 to the field by simply adding it to the formula as shown:





- Add this new load to both solutions.
  - For the optional frequency solution, such a tabular definition is not possible. Instead, a fixed value current load with 1 A can be created.
8. Solve the two (or three) solutions in the same order as before.
  9. Post-processing.  
All results should be very near to those we have analysed in part one of this exercise. (Blank the copper mesh in the post view to compare the results to the first exercise.)
  10. The tutorial is complete. Save your parts and close them.

## 5.5 Resonant Circuit with 3D Coupling

In this tutorial we show how to create a Resonant Circuit in a 1D Simulation. We calculate the circuit current over time and verify the result with a analytic calculation. Later we replace the 1D resistance with a 3D geometry. This concept can be used to connect realistic 3D geometries with 1D circuits.



### 5.5.1 1D Circuit Simulation

1. Start the program NX or Simcenter 3D.
2. In Simcenter, create a new file. Key in for name 'ResonantCircuit1D' and click OK.
3. From toolbar Application click on 'Pre/Post' 
4. Click in toolbar 'Magnetics' on 'New FEM and Simulation (Non-Manifold)' 
5. Create a Solution with type 'Magnetodynamic Frequency'. In 'Output Request', 'Plot', activate 'Electric Potential (phi-Pot)'. In 'Table', activate 'Circuit Voltage' and 'Circuit Current'. As Frequency use 50Hz.
6. In 'Edit Solver Parameters', select 'Create, keep txt Files'.
7. Click 'Node Create' from toolbar 'Nodes and Elements'. Create 4 nodes, approximately in form of a rectangle. Use any positions.
8. Create a '1D Connection' and select with 'Node to Node' the first two nodes. Set the type to 'Resistor'. Create a second 1D Connection, select the next two nodes with type 'Inductor'. Create between the last two nodes another 1D Connection with type 'Conductor'.
9. Edit the physicals of the resistor. Key in  $1\Omega$  for 'Electric Resistance per Element'. Do the same for the 'Inductor' with 1H and 1(Farad) for the 'Capacitance' of the 'Capacitor'.
10. Switch to the Sim File and click 'Load Type', 'Voltage Harmonic'. Switch to 'On Circuit' and select the two open nodes as primary and secondary node. Key in 1V.
11. Solve the Solution.

### 5.5.2 Post Processing

Open the file with the extension '.CurrentCircuit.txt'.

```
ResonantCircuit1D_sim1-Solution_1.CurrentCircuits.txt  
CurrentCircuits_Unit_A CapacitorEle3: 50 -1.013222102405632e-05 0.003183098861511127
```

Shown is the real and imaginary part of the circuit current at every component. Since we have a serial circuit, the current is always the same.

### 5.5.3 Analytic Verification

We set up a differential equation of the 'Serial Resonant Circuit' to calculate the current over the time  $t$ . The differential equation results from the sum of the circuit voltages:

$$U(t) = U_R + U_L + U_C$$
$$U(t) = R \cdot I(t) + L \cdot I'(t) + \frac{1}{C} \cdot \int I(t) dt$$

We derive the equation by time to avoid the calculation with an integral.

$$U'(t) = R \cdot I'(t) + L \cdot I''(t) + \frac{1}{C} \cdot I(t)$$

with  $R = 1\Omega$ ,  $L = 1H$ ,  $C = 1F$  and  $U(t) = \cos(2 \cdot \pi \cdot 50Hz \cdot t)$ , so  $U'(t) = 100\pi \cdot \cos(100\pi \cdot t)$ .

We use the following mathematical approach:

$$I_p(t) = D_0 \cdot \sin(bt) + E_0 \cdot \cos(bt)$$

That approach and its derivations are inserted into the differential equation and after some simplifications we get:

$$\cos(b \cdot t) \cdot b + (\sin(b \cdot t) \cdot 0) = \cos(bt) \cdot (D_0 \cdot b - E_0 \cdot b^2 + E_0) + \sin(bt) \cdot (-E_0 \cdot b - D_0 \cdot b^2 + D_0)$$

with  $b = 100\pi$  and a coefficient comparison two equations with two variables result. Solving the equation system gives

$$D_0 = 1.013 \cdot 10^{-5}, E_0 = -3.18 \cdot 10^{-3}$$

We lead this back to our approach and get a equation describing two oscillations:

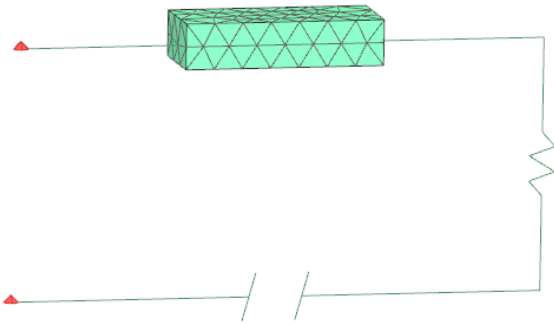
$$I(t) = 1.013 \cdot 10^{-5} \cdot \sin(100\pi \cdot t) - (-3.18 \cdot 10^{-3}) \cdot \cos(100\pi \cdot t)$$

The sum of the two oscillations finally results in


$$I(t) = 3.18A \cdot 10^{-3} \cdot \sin(100\pi \cdot t - 1.571)$$

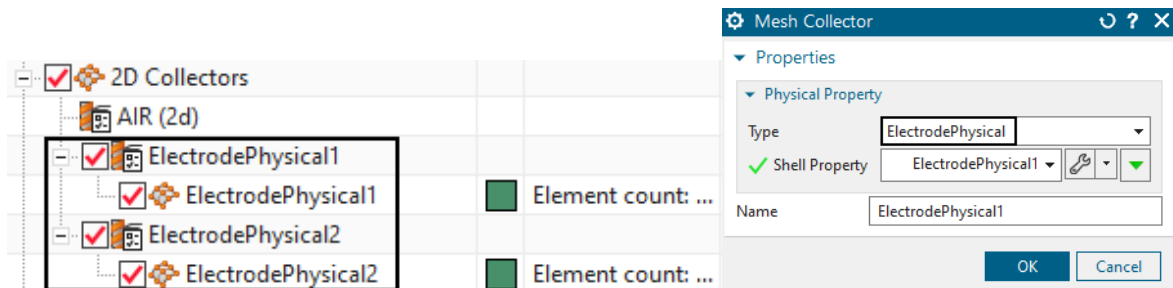
Converting this into a complex number gives  $I = 1.013 \cdot 10^{-5} - j3.18 \cdot 10^{-3}$ . This is the settled result for the current behavior over time and matches precisely with the result of the simulation.

#### 5.5.4 Circuit with 3D Resistance

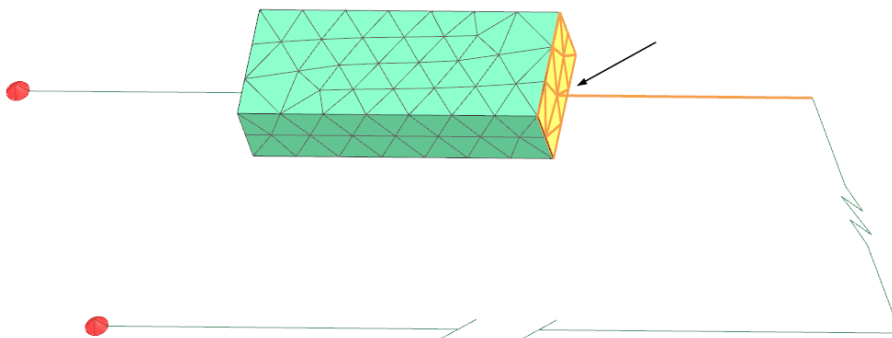


We now replace the 1D Resistor with a 3D geometry. Because of the material properties and the geometry, there arises a resistance. Following we open an existing model and walk through some interesting features.

1. Download the model for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.8ResonantCircuit.zip>
2. Extract the zip-archive. In Simcenter, click Open  and navigate to the folder of the extracted archive. Select the file 'ResonantCircuit3D\_sim1.sim' and click OK.
3. The meshes for the air and the resistor are already created. We use a self made material for the resistor and get a resistance of  $1.07\Omega$  which nearly corresponds to the prior 1D simulation. ( $1.0\Omega$ )
4. There are two 2D meshes of type 'ElectroPhysical'. These are necessary to connect a 1D connection to a 3D geometry. They don't contain any physical properties.



5. To finally connect the 2D electrode with the outside circuit we need a 1D connection of type 'Connector'. Such a connector must be between a node of the electrode and another node in the circuit. See picture below.



6. Switch to the Sim File and solve the solution.
7. Open the file with the extension '.CircuitCurrent.txt'

ResonantCircuit3D\_sim1-Solution\_1.CircuitCurrents.txt

CurrentCircuits\_\_Unit\_A CapacitorEle603: 50 -1.087833047451558e-05 0.003183093918487335

8. Shown is the circuit current, which matches well with the 1D simulation. The small deviation results from the slight difference in the ohm resistance.

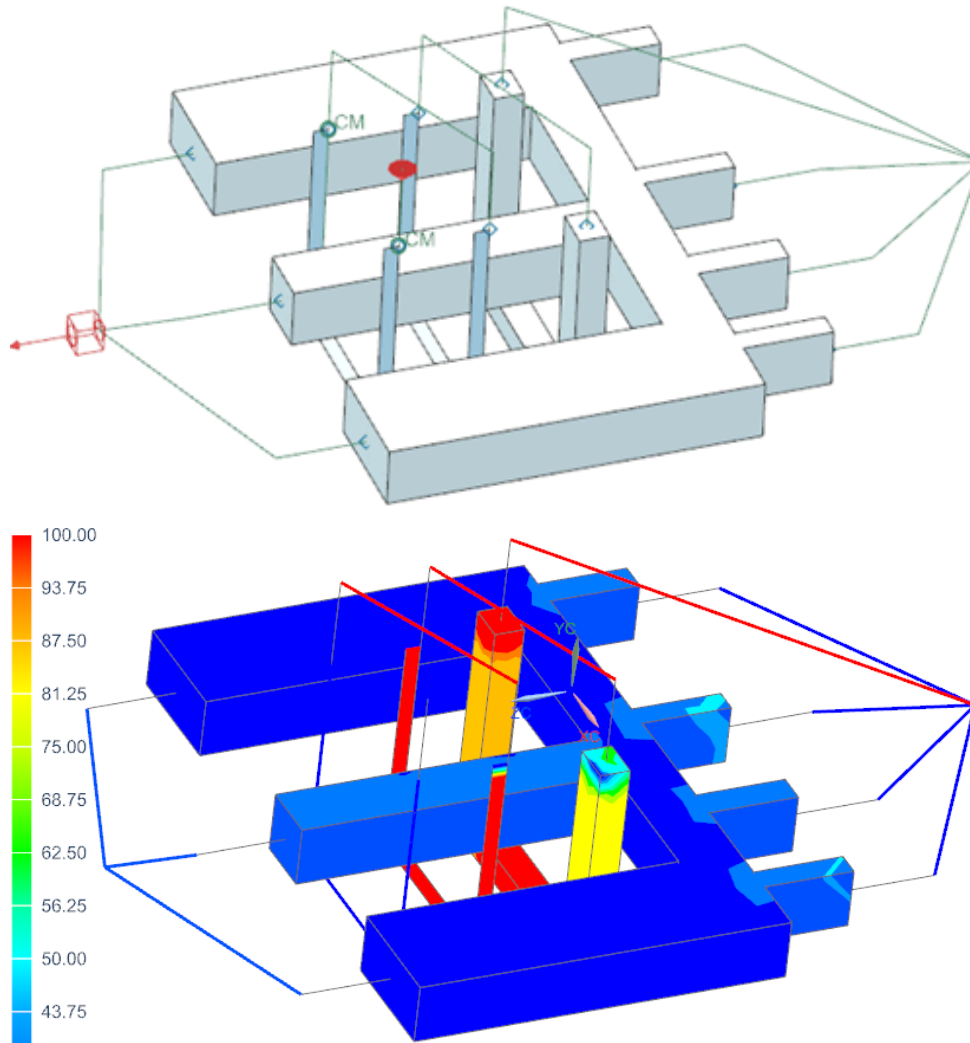
The tutorial is finished.

## 5.6 1D, 2D, 3D Connector

In this example there is a network defined connecting different electrodes of several conductors. The conductors are of 3D, 2D and 1D type. The example shows how to set up these features. A harmonic current load is applied on the left and right points of the network (see picture). So the electric current flows through all conductors.

Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.5Network.zip>



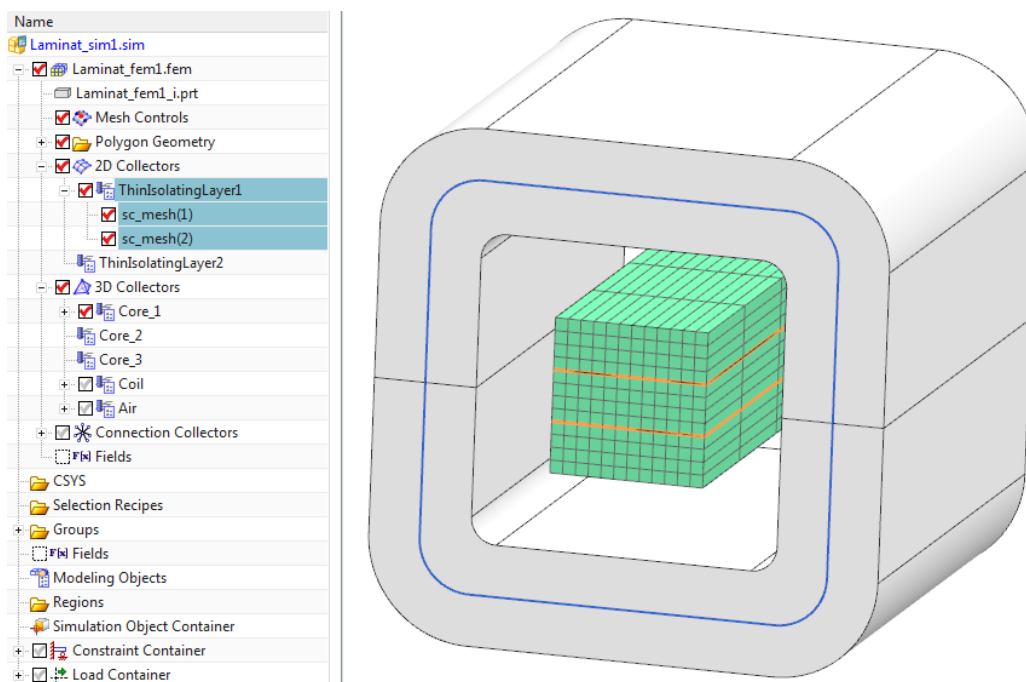
## 5.7 Isolating Layer

In this example there is a core in a coil running in a magnetodynamic simulation. The core has two isolating layers which avoid eddy currents to cross those layers. The example shows how to set up these features.

Download the model files for this tutorial from the following link:

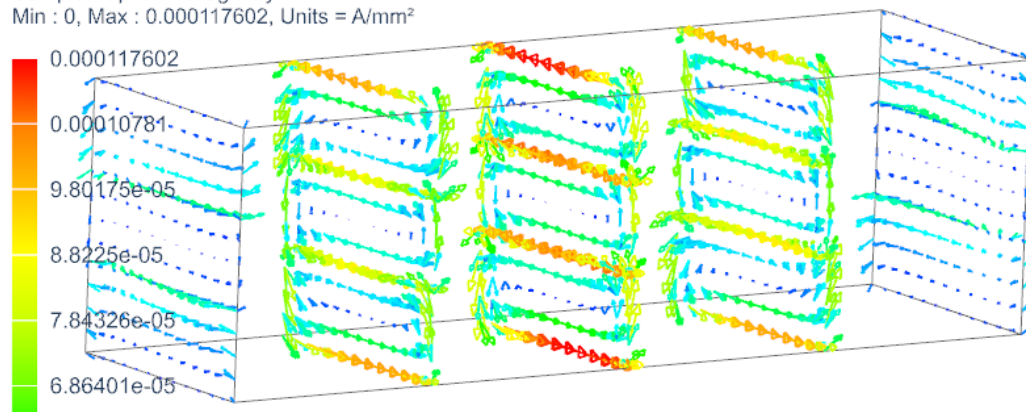
<https://www.magnetics.de/downloads/Tutorials/4.MagDyn/4.6Laminate.zip>

The following figure shows the meshed core and the coil. Two orange sheets (highlighted) are the isolating layers. These are meshed in 2D and assigned to a physical of type IsolatingLayer.



Next figure demonstrates the eddy currents in the core and how they are hindered from crossing the isolating layers.

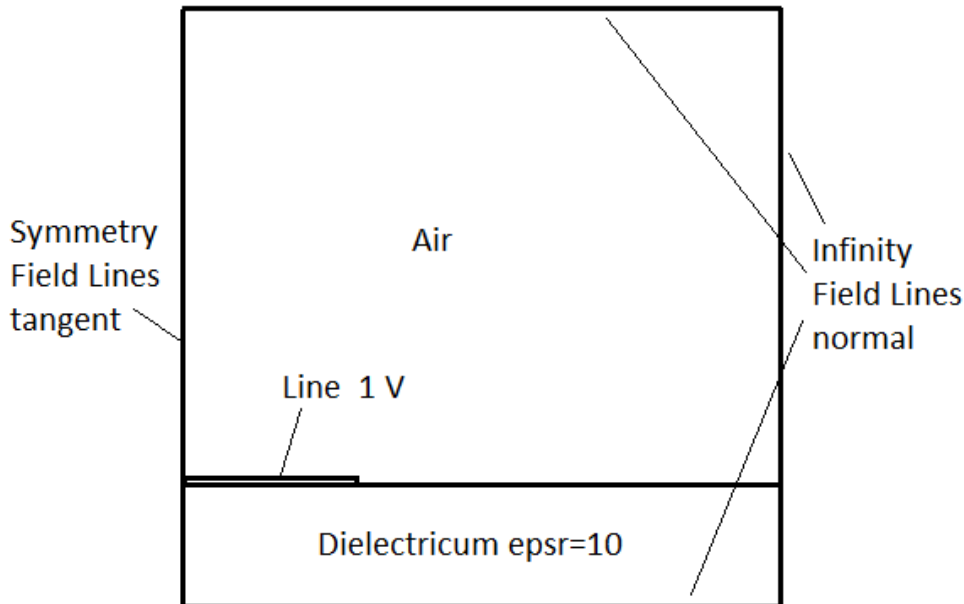
Laminat\_sim1 : Solution 1 Result  
Load Case 1, Frequency 1, 50 Hz  
Current Density (Area) - Element-Nodal, Unaveraged, Magnitude  
Complex Option : Imaginary  
Min : 0, Max : 0.000117602, Units = A/mm<sup>2</sup>



## 6 Electrostatic

### 6.1 Microstrip Line (Beginner)




In this example it is shown how to analyze a simple 2D electrostatic problem. The geometry is a section of a microstrip line. It is symmetric, that's why only one half of the section is modeled. The simulation is possible to set up in 2D or in 3D. Following we work in 2D but both 2D and 3D models are available in the tutorials folder.

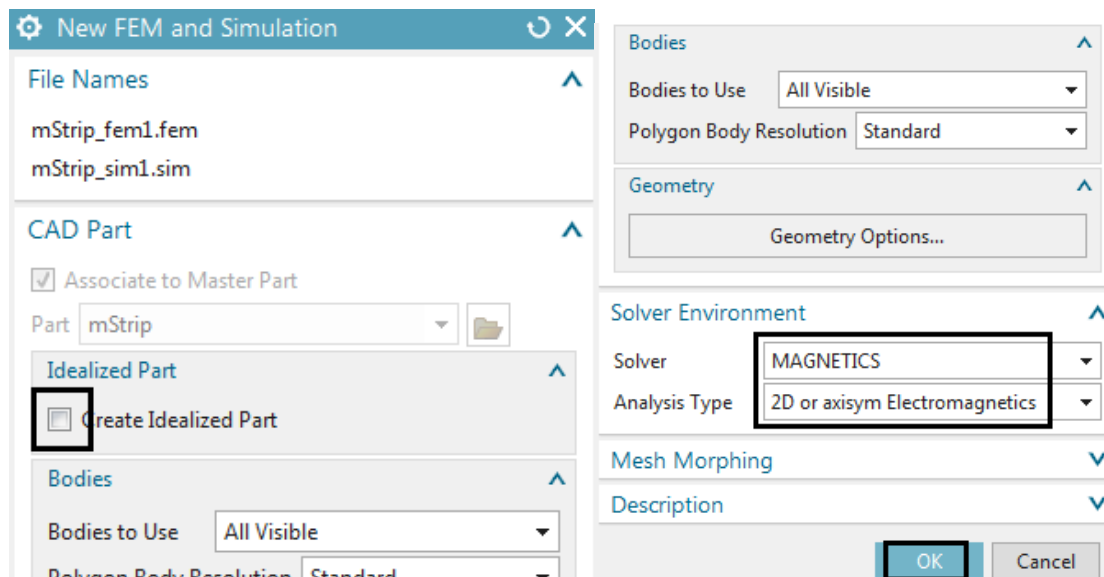


Estimated time: 30 min

Follow these steps in Simcenter

#### 6.1.1 Preparations

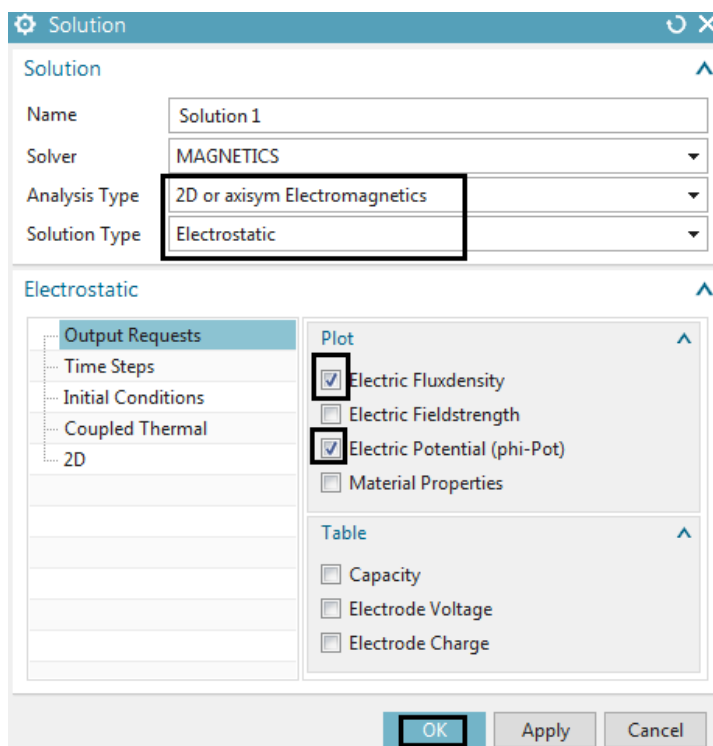
1. download the model files for this tutorial from the following link:  
<https://www.magnetics.de/downloads/Tutorials/1.EleSta/1.1mStrip.zip>
2. unzip the archive. There will be one folder 'start' and one 'complete'.
3. Start the Program Simcenter  (or NX). Use Version 10 or higher.
4. In Simcenter, click Open  and navigate to folder 'start'. Select the file 'mStrip.prt' and click OK. (Maybe you must set the file filter to 'prt')
5. Create New FEM and Simulation 
6. Swith off 'Create Idealized Part' (picture below) because there is no need for doing simplifications.
7. Choose Solver 'MAGNETICS', Analysis Type '2D or axisym Electromagnetics' and click OK.



8. A new window opens to define the solution. Choose Solution Type 'Electrostatic'.

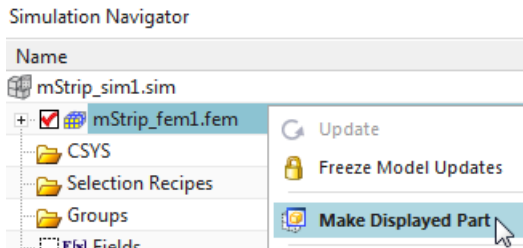
9. In tab 'Output Requests' activate

- 'Electric Fluxdensity' and
- 'Electric Potential (phi-Pot)'
- Click OK to finish the solution definition.





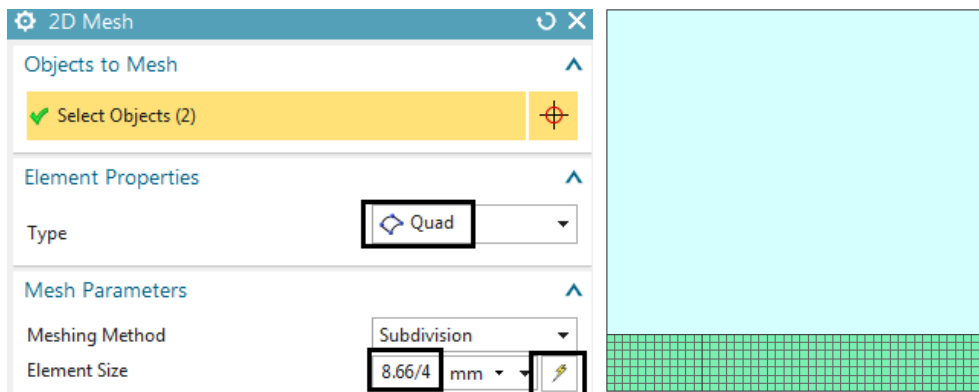
### 6.1.2 Fem File Steps

1. Set the Fem file to the displayed part.




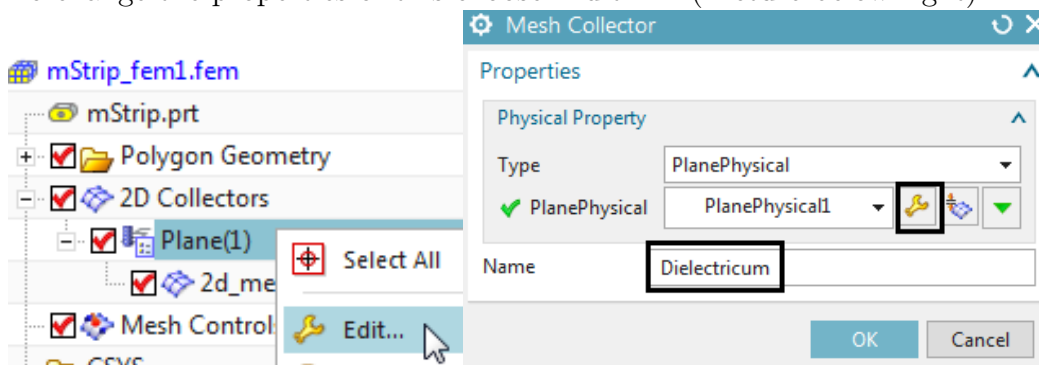
## 2. Mesh the Dielectricum


- Click on '2D Mesh'  and select the two bottom faces under the mStrip line (green in below picture).
- Set the Type to 'Quad'.  
Hint: Tri elements are also possible of course, but this geometry is perfect for Quads.
- Click on 'Automatic Element Size'  to ask the system for a convenient element size. Divide this value by 4 to allow small elements in this area.




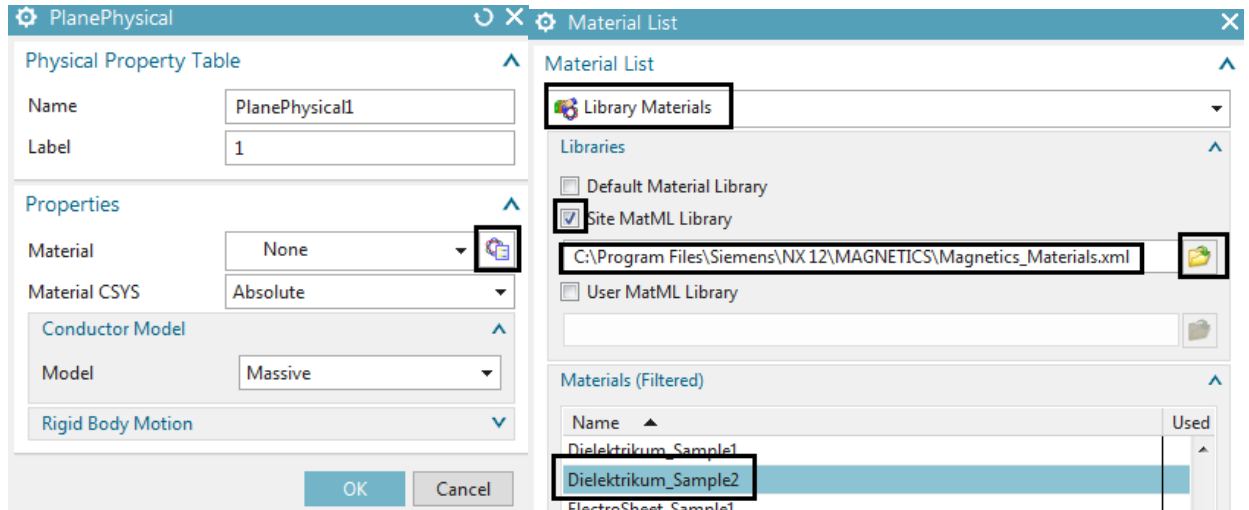
## 3. Assign Properties for the Dielectricum


- Double click the newly created mesh collector 'Plane(1)'. A new window appears. Accept the default type 'PlanePhysical'.
- Change the Name to 'Dielectricum'.
- To change the properties of this choose 'Edit'  (Picture below right)

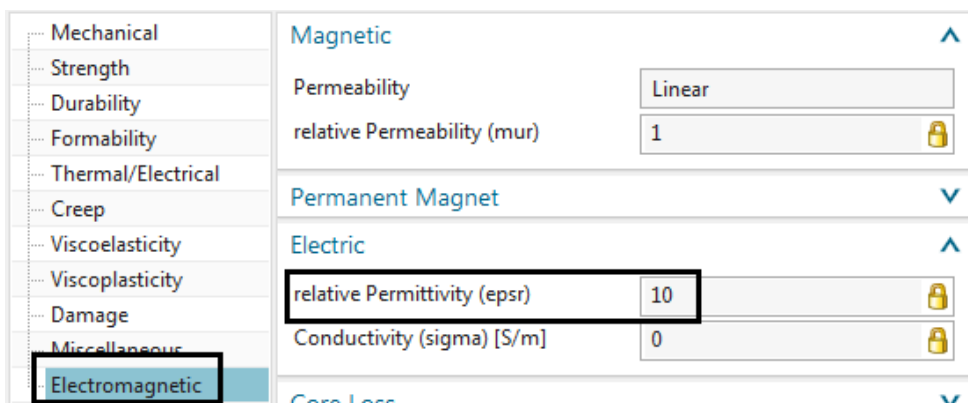


- for assigning the material select the button 'Choose Material'  (Picture below left). A new window appears showing the Material List.

- In the Material List activate 'Library Materials' and 'Site MatML Library'.
- Choose the browse function  and insert the MAGNETICS material library from your installation directory. In a usual Simcenter or NX12 installation this will be something similar to this:  
'.../NX 12.0/MAGNETICS/Magnetics\_Materials.xml'
- From the Material List select the material 'Dielektrikum\_Sample2'





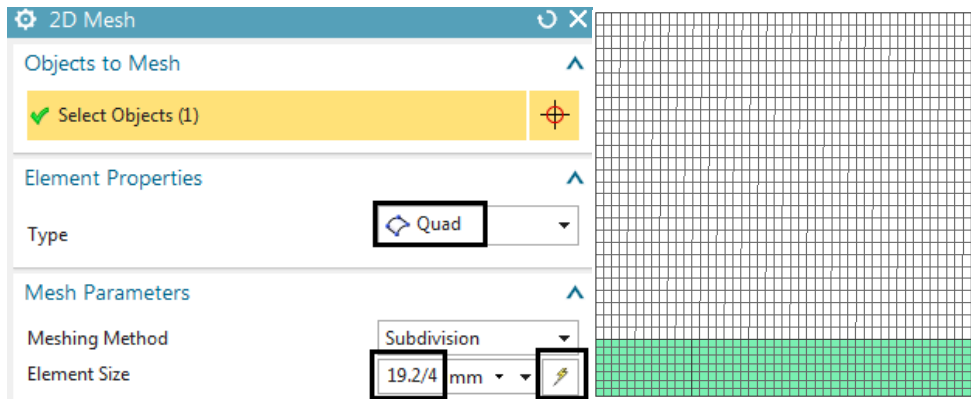
- Hint: To check the properties of the selected material choose 'Inspect Material' . In register 'Electromagnetic' check for the correct setting of the relative Electric Permittivity (epsr):





- Click Close, OK, OK, OK to finish the definition of dielectricum.

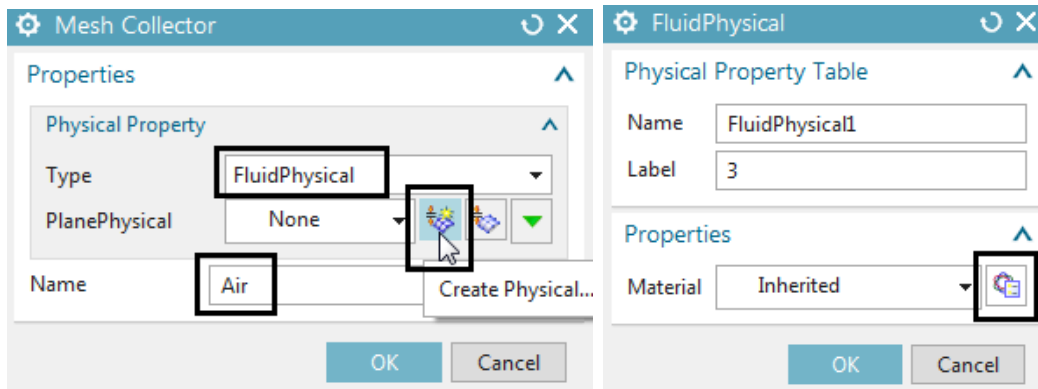
#### 4. Mesh the air:

- Click on '2D Mesh'  and select the upper face.
- Set the Type to 'Quad' and click on 'Automatic Element Size'  to ask the system for a convenient element size. Divide this value by 4.

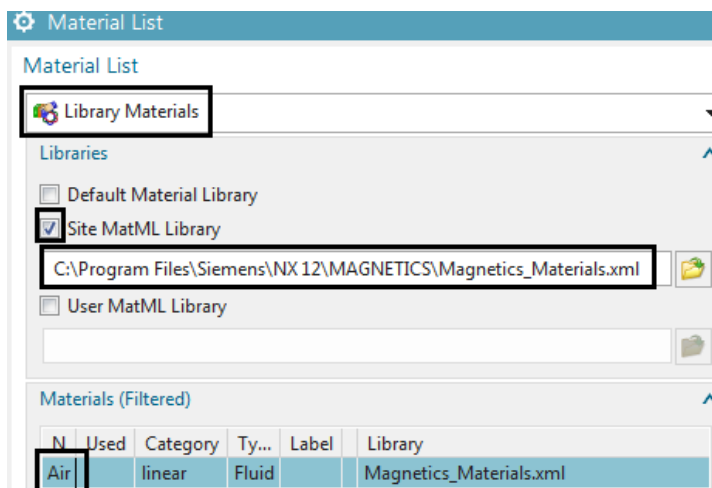


## 5. Assign Properties for Air

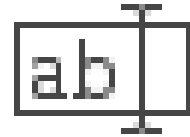
- Edit the newly created mesh collector 'Plane(2)'. Change the Type to 'FluidPhysical' and the Name to 'Air'.
- Then click 'Create Physical'  to create a new physical property for air. In the following window click 'Choose material' .



- Select the material 'Air' from the same library as before.



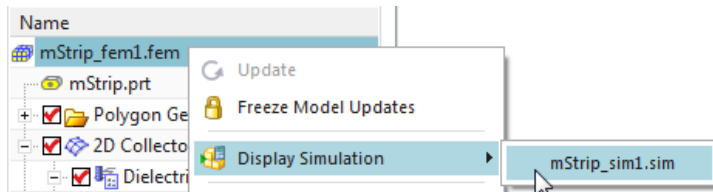
- Click Close, OK, OK to finish the definition of air.



6. For easier post processing, click on the button 'Rename Meshes ...' from the Magnetics toolbar. That utility renames all meshes and therefore is a useful feature especially with larger models and many meshes.

### 6.1.3 Sim File Steps

1. Switch to the Sim file



2. Hint: Possible Constraints in Electrostatics are

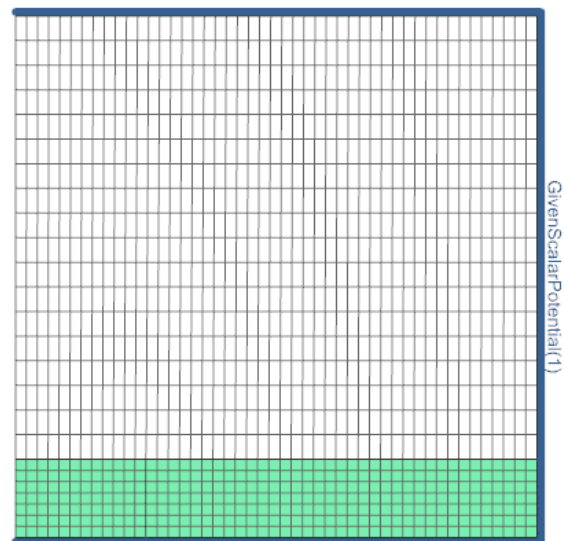
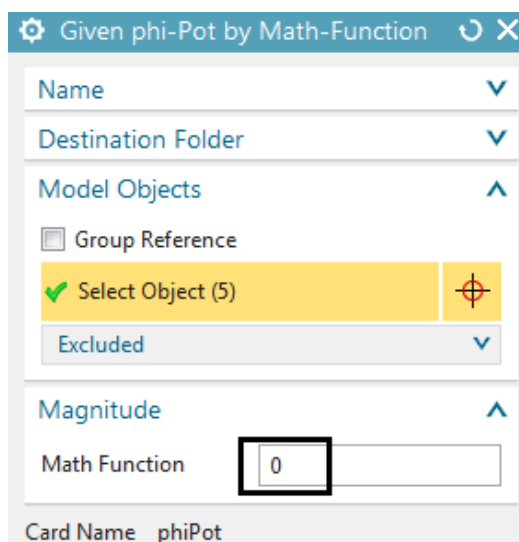
- Given phi-Pot by Math-Function  
Use this function to assign a mathematical function or simply a value for Voltage
- Perfect Conductivity Boundary  
This condition simulates the behavior of a perfect conductor e.g. there will be a constant voltage. The magnitude of this voltage will be calculated by MAGNETICS

3. Hint: Possible Loads are

- Charge [ $A*s$ ]

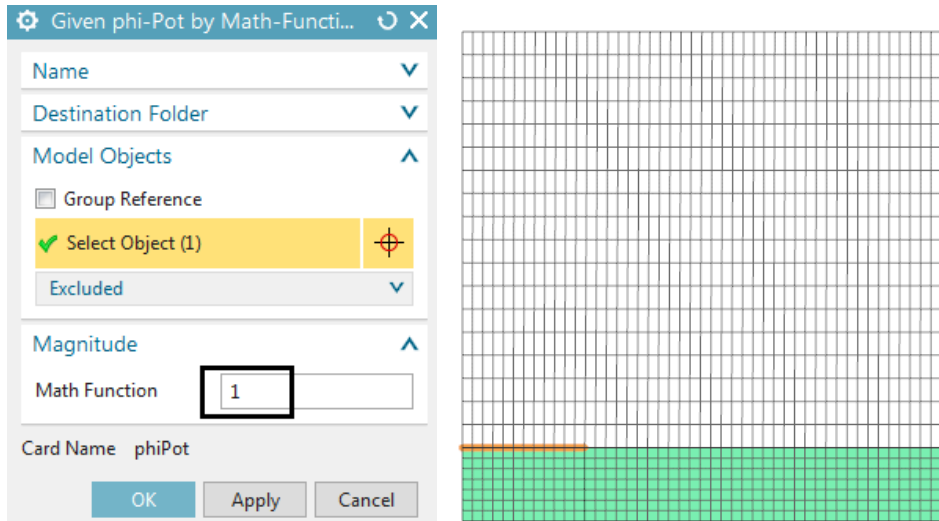
4. To create a constraint that results in normal field lines at the outside edges:

- Click on 'Given phi-Pot by Math-Function'  $f(x)$  and select the 5 outside edges, but not the symmetry edges (see picture below right).
- Assign a Magnitude of 0. Click OK.



5. To assign the constraint '1 V' on the microstrip line (see picture below):

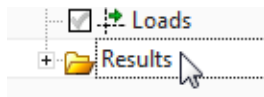
- Click on 'Given phi-Pot by Math-Function'  $f(x)$  and select the microstrip edge.
- Assign a Magnitude of 1. Click OK.



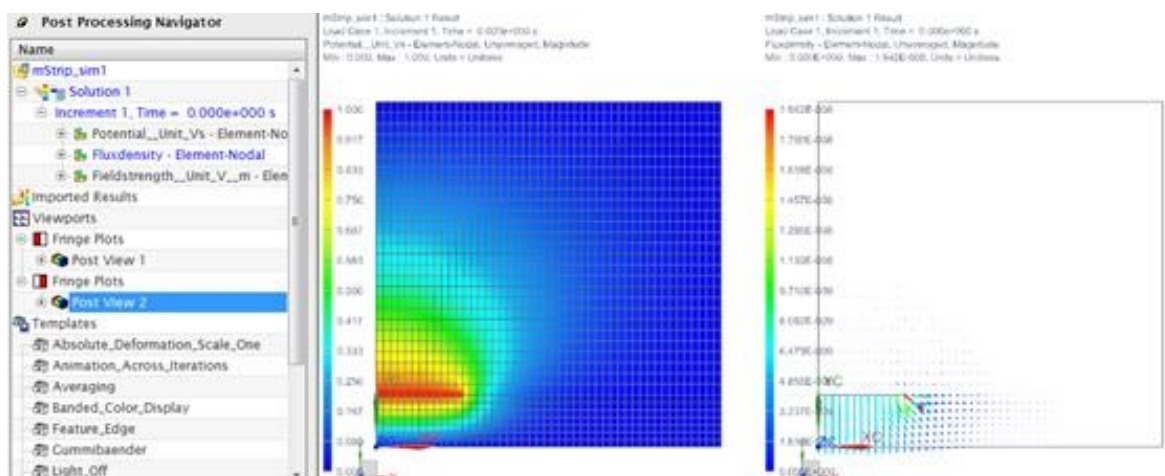
6. Solve the Solution: Click on 'Solve'  $\begin{matrix} + \\ - \end{matrix} =$  and OK.

7. Perform Post Processing:

- Double click the Result. The Simcenter system switches to the Post Processing Navigator.



- In the Post Processing Navigator double click the Electric Potential Result or any other to see its field plot.
- The picture below shows on the left side the electric potential and on the right side the electric flux density result with arrows.



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

## 7 Electrokinetic

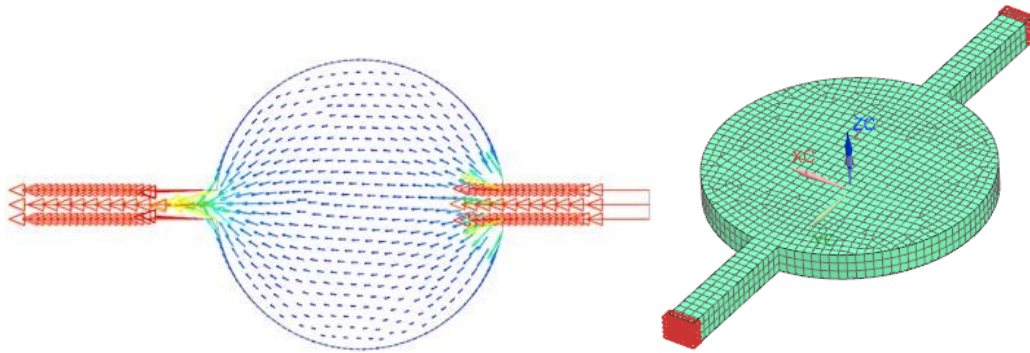
### 7.1 Circular Plate

Goal of this analysis is to analyze the electric current flow in a circular plate to compute its ohm resistance.



Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/2.EleKin/2.1CircularPlate.zip>

This model already contains all needed properties and parameters.

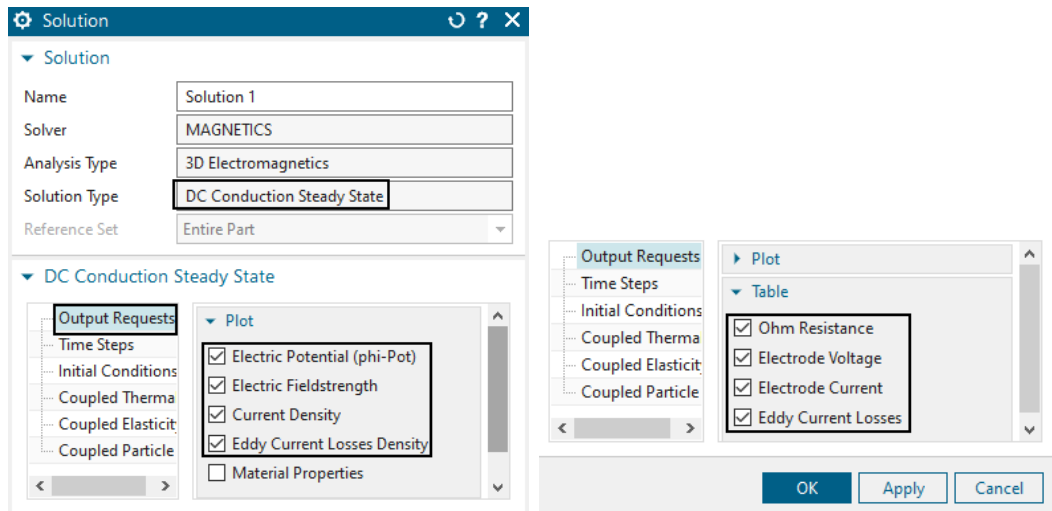


#### 7.1.1 Preparation

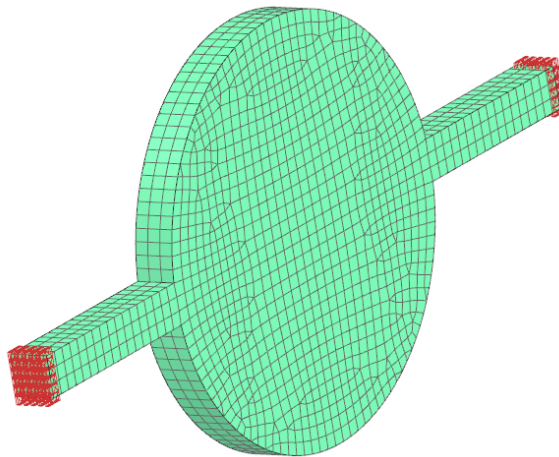
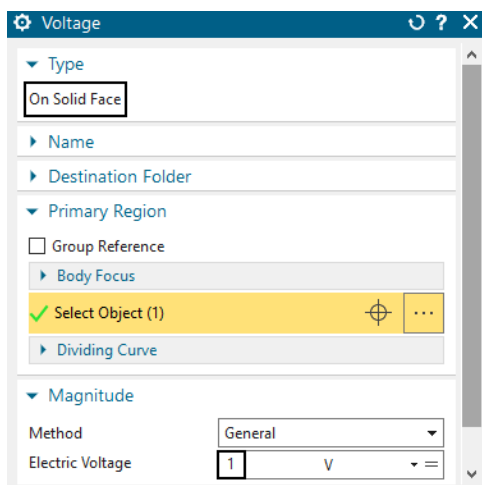
1. Unzip the archive. There will be one folder 'start' and one 'complete'.
2. Start the Program Simcenter  (or NX).
3. In Simcenter, click Open  and navigate to folder 'complete'. Select the file 'CircularPlate\_sim1.sim' and click OK. (Maybe you must set the file filter to 'sim')
4. Click from toolbar 'Application', 'Pre/Post'.

#### 7.1.2 Sim File Check

1. Edit the already existing Solution 'Solution1', to check the properties.
  - The 'Analysis Type' is '3D Electromagnetics', the 'Solution Type' is set to 'DC Conduction Steady State'.  
Hint: This solution type simulates the electric field in electric conductors and the resulting electric current. The magnetic field is not considered. From the electric current, the geometry and the boundary conditions, the ohm resistance can be derived.
  - Under 'Output Requests' 'Plot', 'Electric Potential (phi-Pot)', 'Electric Fieldstrength', 'Current Density' and 'Eddy Current Losses Density' are active to check these results.
  - In 'Table', 'Ohm Resistance' and some others are active.



2. The 'Result Graphs (afu)' in 'Edit Solver Parameters' is set to 'Create, keep txt Files'.
3. Edit the load 'Voltage(1)'. The type is 'On Solid Face' and the face of the supply line is selected. The 'Electric Voltage' is set to 0V.
4. Edit the second load. The other side of the supply line is selected and the 'Electrode Voltage' is set to 1V.



5. Make the Fem File to the displayed part.
6. In '3D Collectors', edit the mesh collector 'Copper'. Edit the physical and verify the material is set to copper.
7. Check the properties of the 3d mesh. In this case there is a hex mesh used. The use of tet mesh is also possible.
8. Solve the Solution

### 7.1.3 Post Processing

1. Open the File with the extension '.Resistance.txt'.  
The calculated resistance of the full geometry circuit plate is shown.

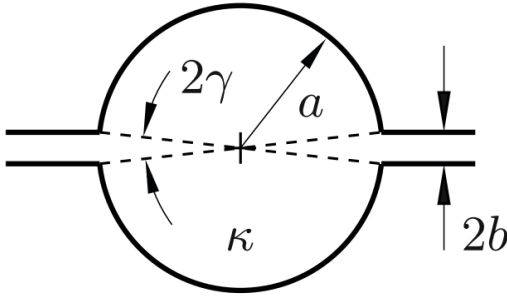
CircularPlate_sim1-Solution_1.Resistance.txt						
1	Resistance	Unit_Ohm	Voltage_1	0	0	Voltage_2
				0	1.065957405448252e-05	

#### 7.1.4 Result Comparison with Analytic

For analytic calculation, we use a substitute resistance to calculate the ohm resistance of the circular plate.

$$R_0 = \frac{2a}{2bd\kappa} = \frac{0.2m}{2 \cdot 10mm \cdot 20mm \cdot 58S/m} = 8.6207 \cdot 10^{-6}\Omega$$

with  $a = 100mm$ ,  $b = 10mm$ ,  $d = 20mm$ ,  $\kappa = 58S/m$



Next we have to determine the resistance of the circular plate depended on the entrance angle  $\gamma$ .

$$\gamma = \sin^{-1} \cdot \left(\frac{a}{b}\right) = 0.10016[rad]$$

The ratio between the resistance of the circular plate  $R$  and  $R_0$  is:

$$\frac{R}{R_0} = \frac{2\gamma}{\pi} \cdot (1 - \ln(\frac{\gamma}{2})) = 0.2543$$

So, the resistance of the plate  $R$  is:

$$R = \frac{2\gamma}{\pi} \cdot (1 - \ln(\frac{\gamma}{2})) \cdot R_0 = 0.2543 \cdot 8.6207 \cdot 10^{-6}\Omega = 2.193 \cdot 10^{-6}\Omega$$

The resistance for the supply line is calculated by:

$$R_{sup} = \frac{a}{2bd\kappa} = \frac{0.2m}{2 \cdot 10mm \cdot 20mm \cdot 58S/m} = 4.3103 \cdot 10^{-6}\Omega$$

The combined resistance is:

$$2 \cdot R_{sup} + R = 2 \cdot 4.3103 \cdot 10^{-6}\Omega + 2.193 \cdot 10^{-6}\Omega = 1.081 \cdot 10^{-5}\Omega$$

Source: Manfred Filtz, Heino Henke. Übungsbuch Elektromagnetischer Felder. Berlin: Springer Berlin Heidelberg, 2007

- Simulation Result:  $1.066 \cdot 10^{-5}\Omega$
- Analytic Result:  $1.081 \cdot 10^{-5}\Omega$
- Deviation: 1.4%

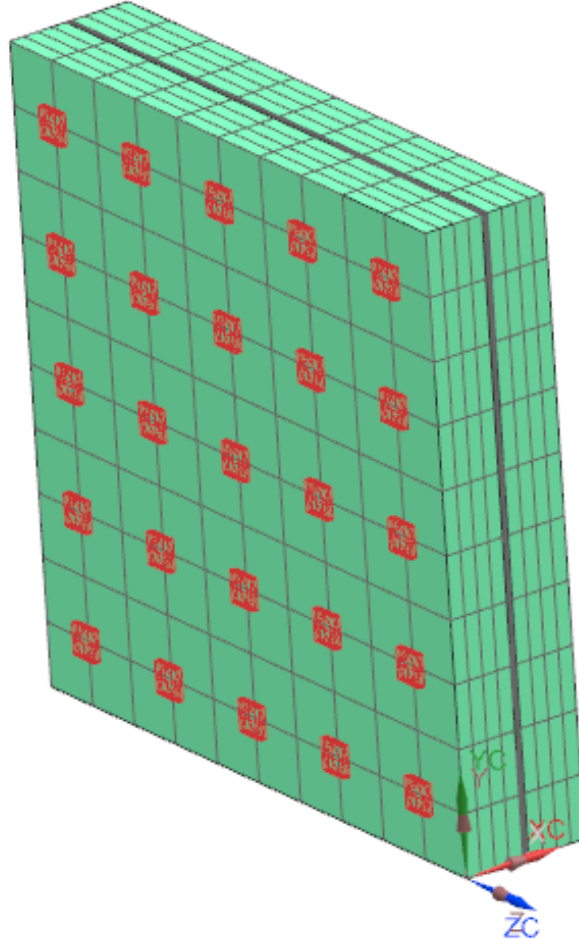
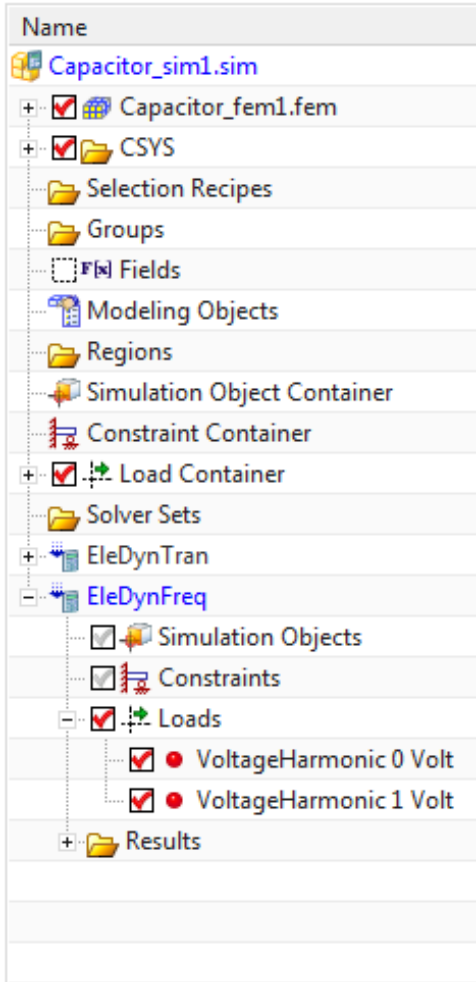
## 8 Electrodynamic

### 8.1 Capacitor

Goal of this analysis is to analyze the electric field in a capacitor.

Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/9.EleDyn/9.1Capacitor.zip>



#### 8.1.1 Result Comparison with Analytic

The capacity  $C$  of a capacitor with area  $A$  and distance  $D$  can be calculated from

$$C = \epsilon_0 \cdot \epsilon_r \cdot \frac{A}{D} = 88.5pF$$

with  $\epsilon_0 = 8.854187 \cdot 10^{-12}$  electric constant,  $\epsilon_r = 1$  material property,  $A = 1 \cdot 10^{-4}m^2$ ,  $D = 1 \cdot 10^{-5}m$

The electric current  $I$  in a capacitor circuit with a frequency  $f$  and capacity  $C$  results from

$$I = -j \cdot 2\pi \cdot f \cdot C = -j \cdot 2.781624 \cdot 10^{-8}A$$

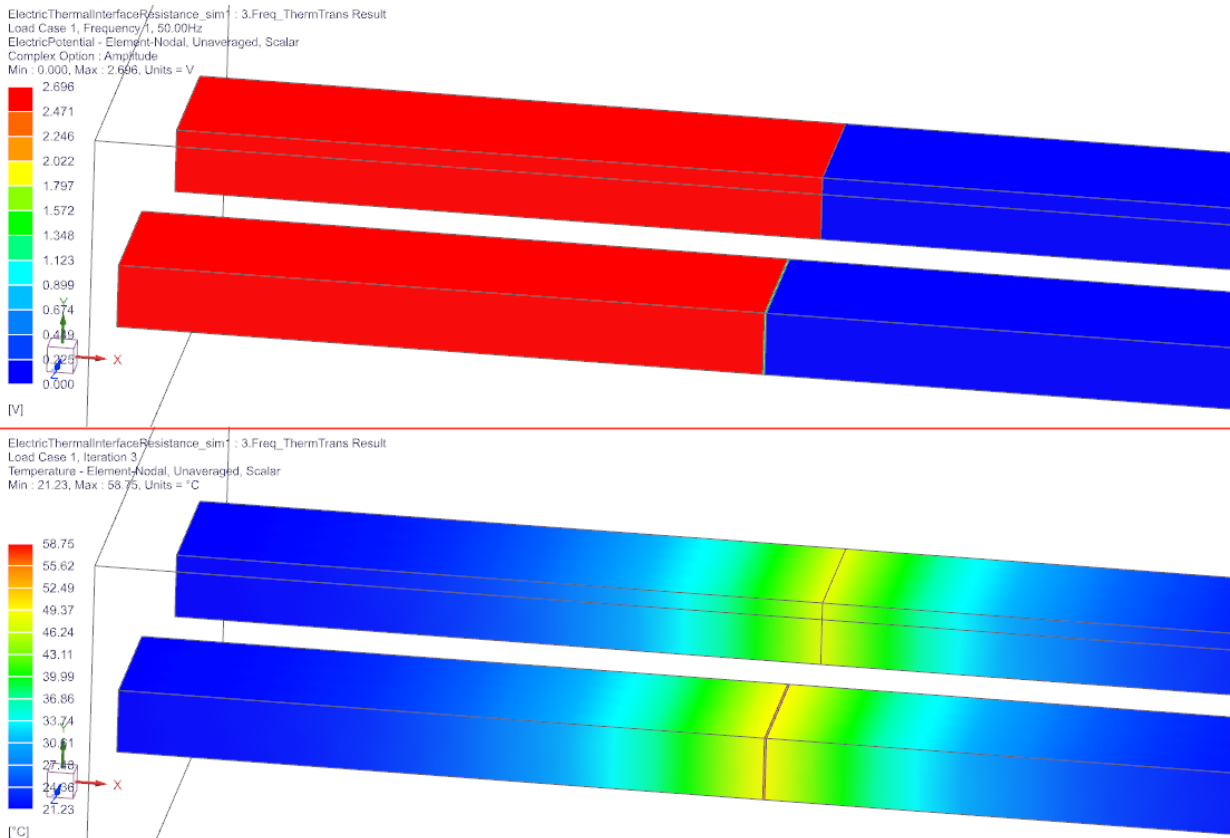
The simulation result is  $2.781624476936687 \cdot 10^{-08}$  and thus, the deviation is smaller than 0.1%.

## 9 More Features

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

## 9.1 Contact Resistance - Electrical and Thermal

In this example we show the feature 'Contact Resistance (One Surface)', that can be used to model thin gaps. For comparison, the simulation model contains two conductors with identical boundary conditions. They differ only in the model of the gap: One has a gap with 3D elements (e.g. the conventional way) while the other uses the feature 'Contact Resistance (One Surface)' with 2D elements at the gap. Both results are very similar. The below picture shows the two conductors: In front the one with 3D, behind the one with 2D elements at the gap. The gap has 0.25 mm thickness, thus it is hardly visible. The upper shows electric potential, the below shows temperature results.



Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/10.Features/10.6ElectricThermalInterfaceRes.zip>

On the left and right electrode faces there are fixed currents (100 Amps) and zero voltages applied. Different solution types are used:

- DC Conduction - steady State
- Magnetodynamic Frequency with coupled Thermal (Static)
- Magnetodynamic Frequency with coupled Thermal (Transient)
- other solution types are also possible.

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

**Contact Resistance (one Surface)**

▶ Name

▶ Destination Folder

▼ Geometry

✓ Interface Region: Region1

Gap Thickness: 0.25 mm

▼ Electric

✓ Effect on Electric Field

Electric Conductivity: 58 S/m

Relative Electric Permittivity: 1

▼ Magnetic

✓ Effect on Magnetic Field

Relative Magnetic Permeability: 1

▼ Thermal

✓ Effect on Thermal Field

Thermal Conductivity (K): 3.87 W/(m·K)

Specific Heat (CP): 385000000 μJ/(kg·K)

Mass Density (RHO): 8.92e-06 kg/mm³

▼ Elasticity

□ Effect on Elasticity

□ External Contact Pressure Field affects Gap Conductivities

Card Name: ContactResistance

The text file 'ElectricThermalInterfaceResistance\_sim1-1.DC.EddyCurrentLoss.txt' contains the computed eddy current losses. It can be seen that the losses in the 3D and those in the 2D region, are very close:

- Eddy Current losses in 3D gap: 134.6982758915086 W.
- Eddy Current losses in 2D gap: 134.6982758297983 W.

The tutorial is complete.

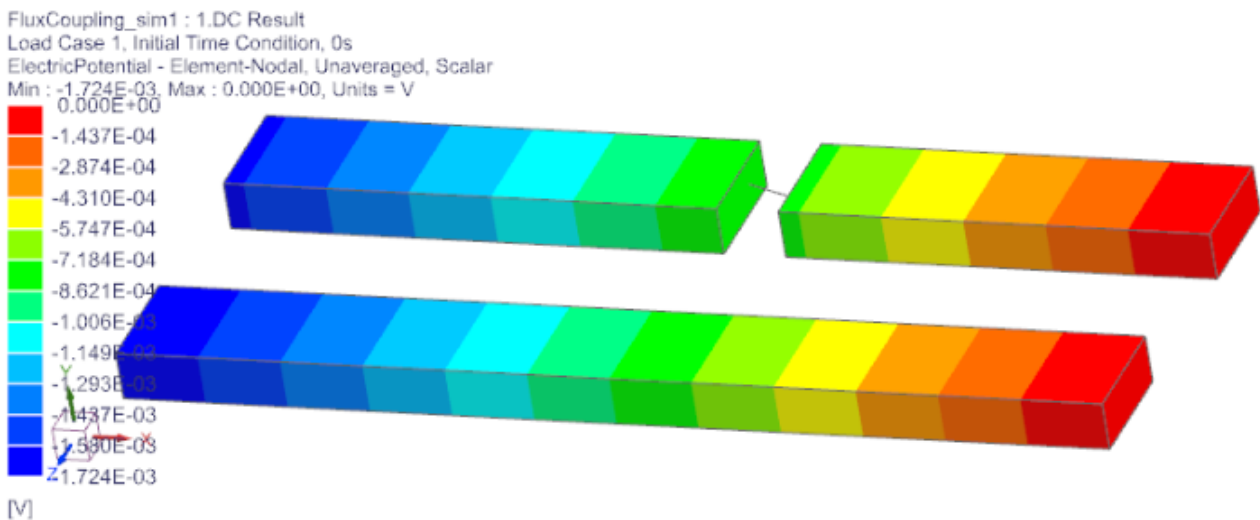
## 9.2 Flux Coupling - Electrical and Thermal

In this example we show the feature 'Flux Coupling (Surface-to-Surface)' that can be used to couple the flux from two faces. Either the electric or the thermal or both fields can be coupled and the two faces must not match or use the same nodes. The feature creates a bidirectional circuit link between the two faces that allows the flux to be coupled. For comparison, the simulation model contains two conductors with identical boundary conditions. They differ only in the model of the gap: One has a gap with 3D elements (e.g. the conventional way) while the other uses the feature 'Flux Coupling (Surface-to-Surface)'. The below picture shows the two conductors: In front the one with 3D, behind the one with the Flux Coupling.

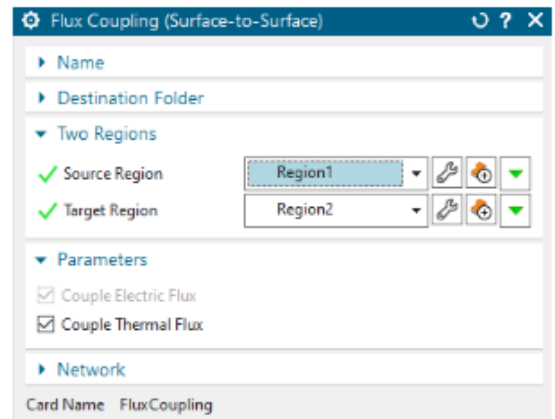
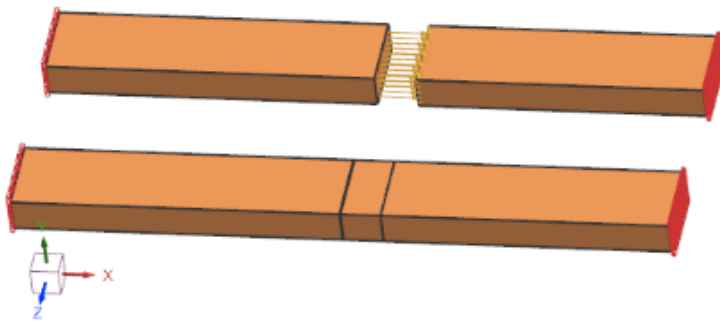
Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/10.Features/10.7FluxCoupling.zip>

There is one model (FluxCoupling\_sim1.sim) with a DC and a Frequency solution. The other model (Flux-Coupling\_sim2.sim) contains also thermal solutions and uses 'Flux Couplings' with additional 'Contact Resistance'. The following figure shows the two conductors. On the left and right electrode faces there are fixed currents of 100 Amps and zero voltages applied. Thus, an electric current will flow through the conductors



The solution is a DC Conduction, but also other solution types can be used. If the faces are inside of a fluid, be aware that the fluid must not touch the faces. The example model shows how that can be achieved. The picture below shows the dialogue of the simulation object that is used to create such an 'Flux Coupling (Surface-to-Surface)'



The tutorial is complete.

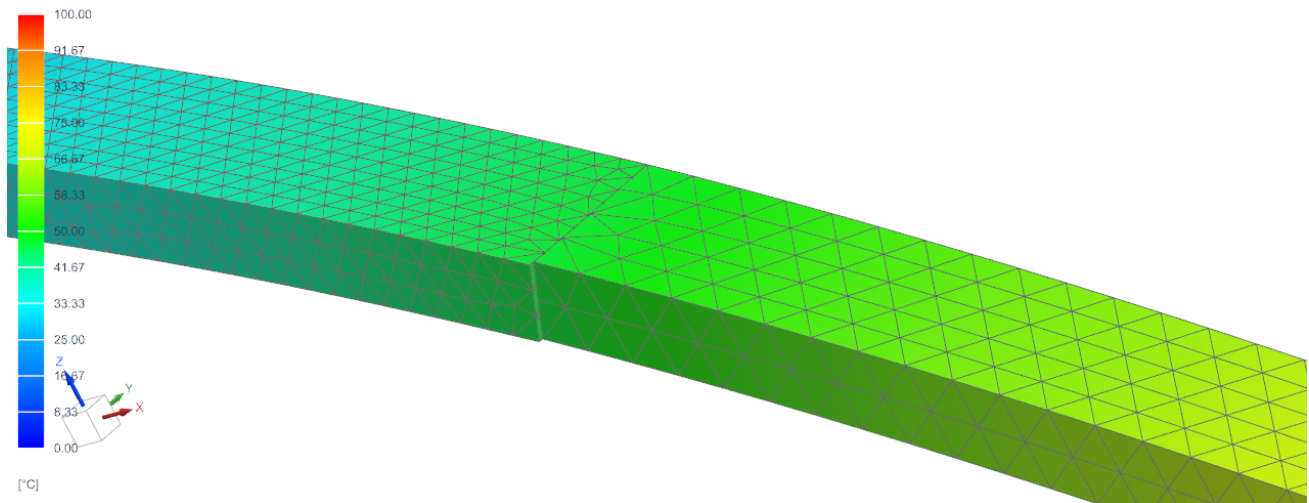
## 9.3 Surface to Surface Glue - Electromagnetic, Elastic and Thermal

In this example we show the feature 'Surface to Surface Glue' that can be used to couple two faces with non-conformal mesh, e.g. with non-matching nodes. The feature uses the mortar FEM method that allows the faces to be coupled.

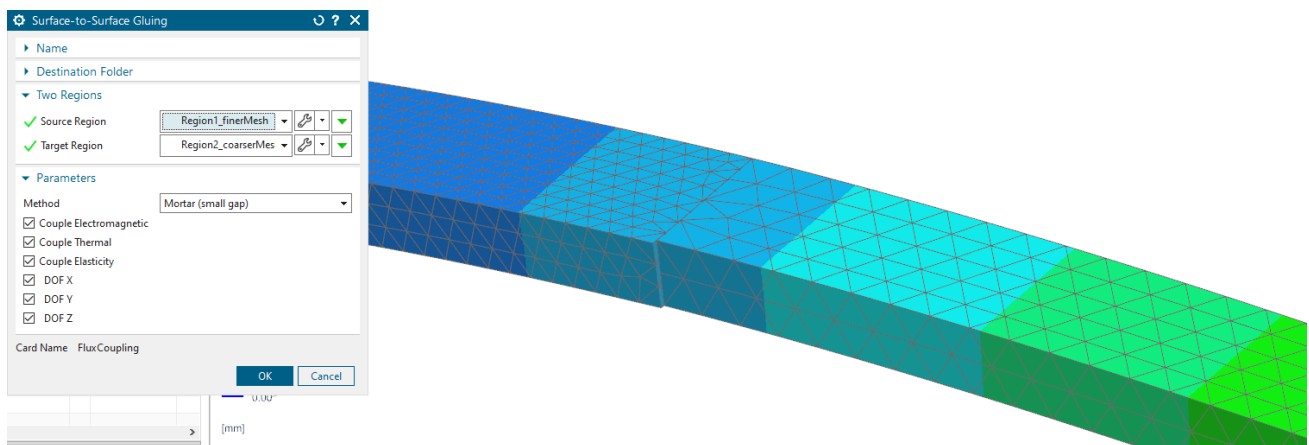
Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/10.Features/10.8SurfaceSurfaceGlue.zip>

There is a model (MortarGlue\_sim1.sim) with a Elasticity and a Thermal solution. The following figure shows the conductor. On the left and right electrode faces there are fixed Temperatures of 0 and 100 deg.



The solution is a DC Conduction, but also other solution types can be used. If the faces are inside of a fluid, be aware that the fluid must not touch the faces. The example model shows how that can be achieved. The picture below shows the dialogue of the simulation object that is used to create such an 'Surface-to-Surface Glue'



The tutorial is complete.

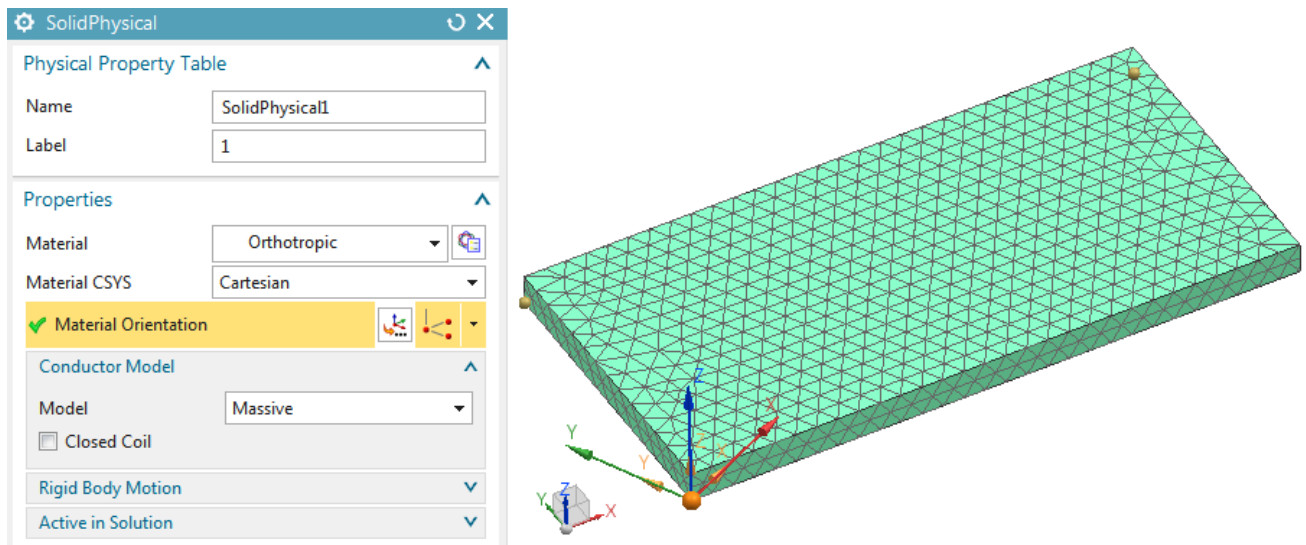
## 9.4 Orthotropic Materials Modeling

In this example electric current flows through a conductor that is modeled with orthotropic electric conductivity.

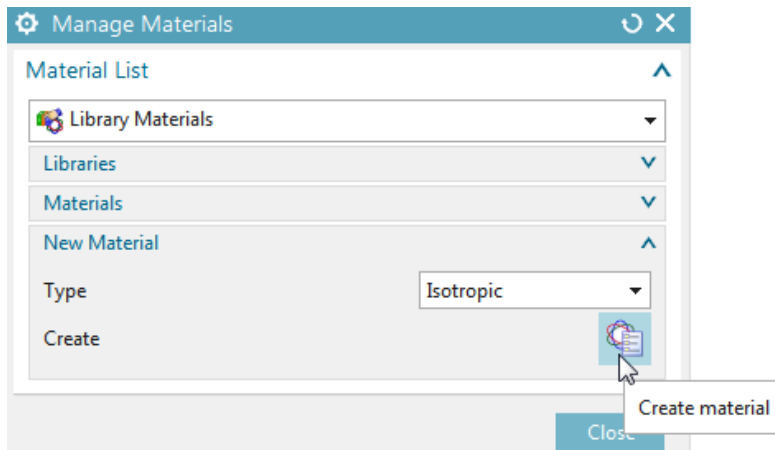
Download the model files for this tutorial from the following link:

<https://www.magnetics.de/downloads/Tutorials/10.Features/10.2OrthotropicMaterial.zip>

The following figure shows the conductor and his physical properties menu. Notice the Material CSYS is set to Cartesian and a new coordinate system is defined with x direction pointing diagonal across the plate.



An orthotropic material is created as shown in the next picture.



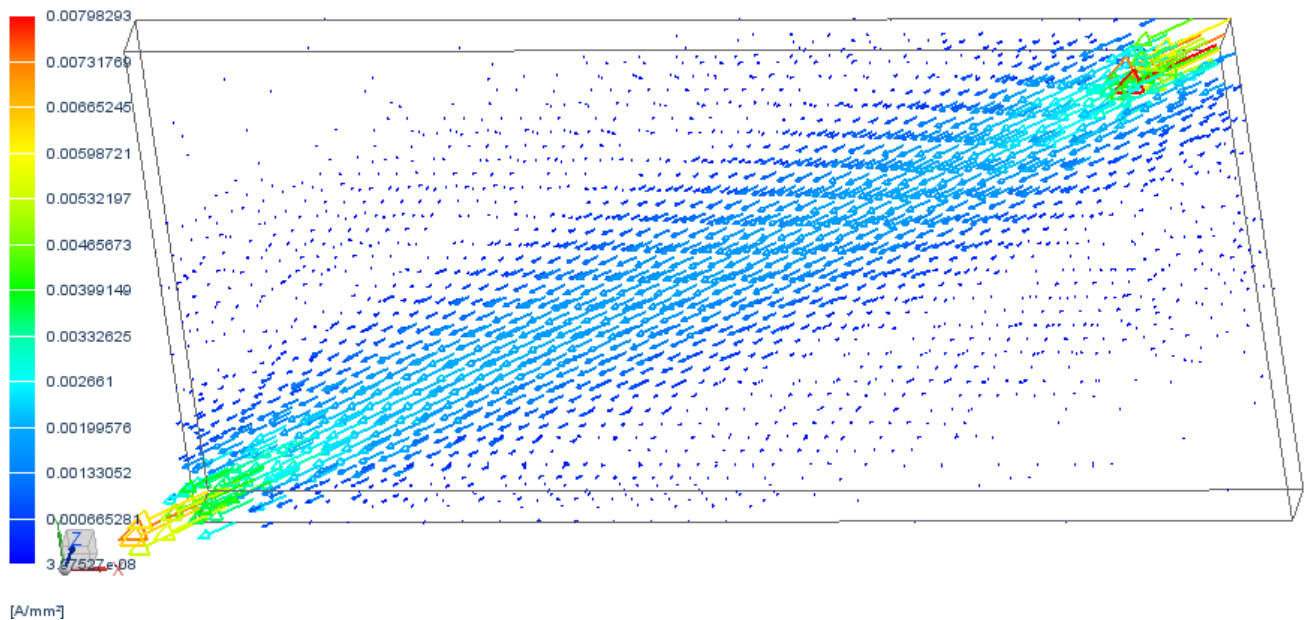
The properties of this orthotropic material are shown below. In this case we use orthotropic electric conductivities and we set the x value much higher than y and z.

Mechanical	Magnetic	▼
Strength	Permanent Magnet	▼
Durability	Electric	▲
Thermal/Electrical		
Creep	relative Permittivity X (epsr)	1 =
Viscoelasticity	relative Permittivity Y (epsr2)	1 =
Viscoplasticity	relative Permittivity Z (epsr3)	1 =
Damage	Conductivity X (sigma) [S/m]	10000 =
Miscellaneous	Conductivity Y (sigma2) [S/m]	10 =
Electromagnetic	Conductivity Z (sigma3) [S/m]	10 =

On the left and right electrode faces there are loads (one ampere left and zero volt right) applied and a DC Conduction Steady State analysis is performed.

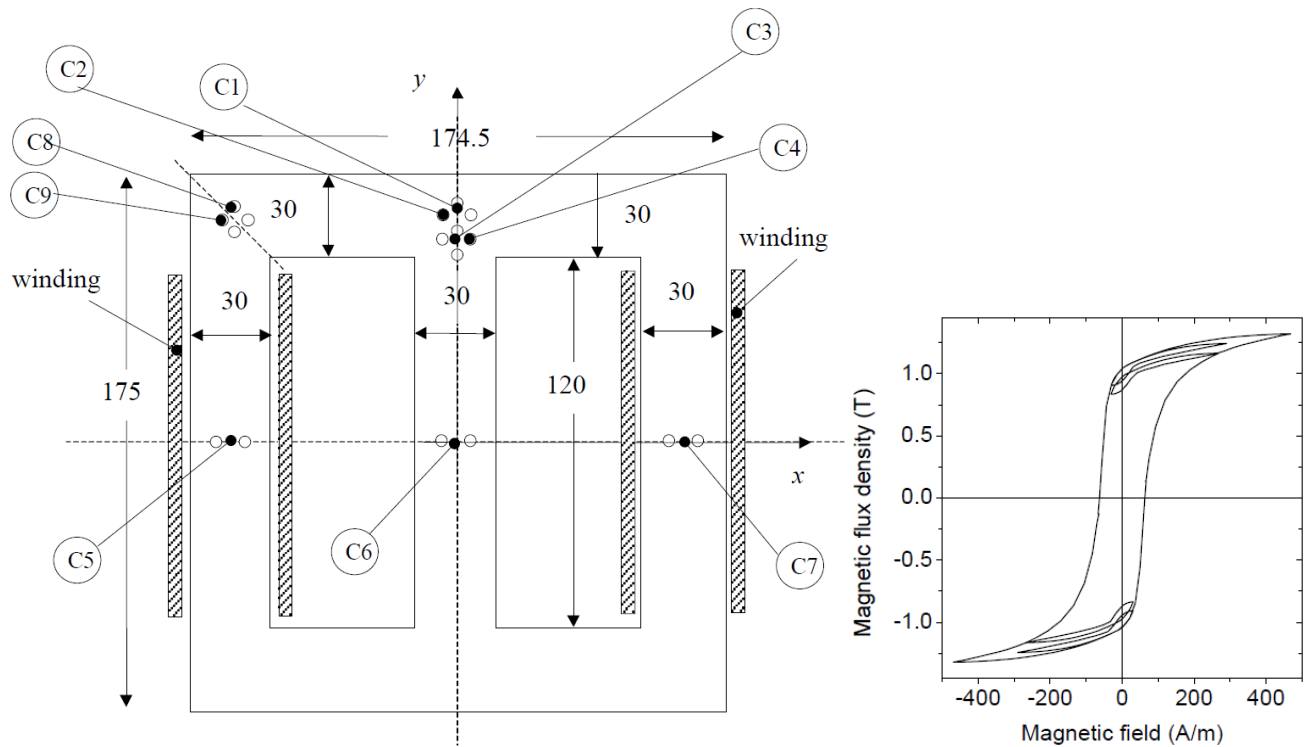
The next figure shows the electric current density as it results from this simulation.

OrthoPlate\_sim1 : DC Result  
 Load Case 1, Increment 1, 0 s  
 Current Density (Area) - Element-Nodal, Averaged, Magnitude  
 Min : 3.97527e-08, Max : 0.00798293, Units = A/mm<sup>2</sup>



## 9.5 Magnetic Hysteresis, Jiles-Atherton Model

In this example we model magnetic hysteresis effects using the Jiles-Atherton model. The used example is the TEAM 32 test case whose setup is shown in the below left picture. The picture right side shows the expected hysteresis loop as it results from alternating magnetic fields at point C6.



More background information about this test example and available measurements can be found in the reference paper at [www.compumag.org/wp/wp-content/uploads/2018/06/problem32.pdf](http://www.compumag.org/wp/wp-content/uploads/2018/06/problem32.pdf)

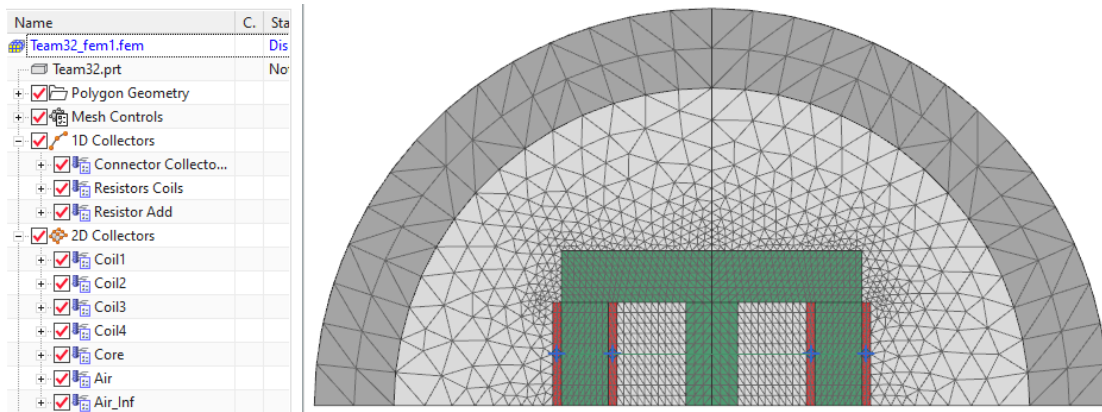
The model consists of two coils and a core. In the paper there are 4 cases described. We model case 2: The two coils are connected in series. The series is supplied by a sinusoidal voltage of 13.5 V (peak value). Additionally there is a fifth harmonic applied with same phase.

Download the model files for this tutorial from the following link:

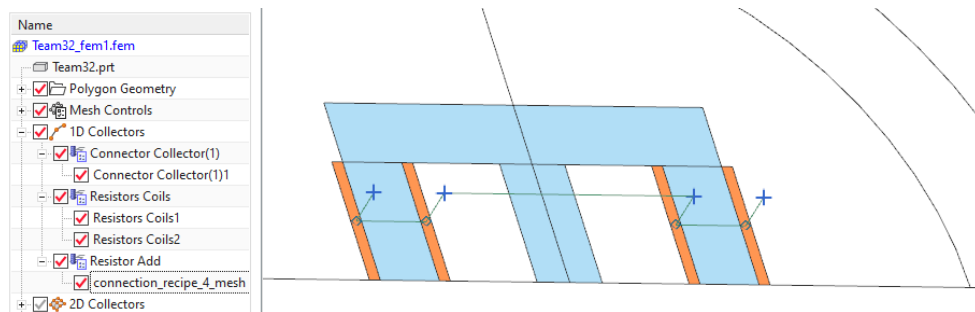
<https://www.magnetics.de/downloads/Tutorials/10.Features/10.5MagHysteresisJilesAtherton.zip>

The model is already build and ready to solve in subfolder complete. Following we walk through some interesting parts of the model.

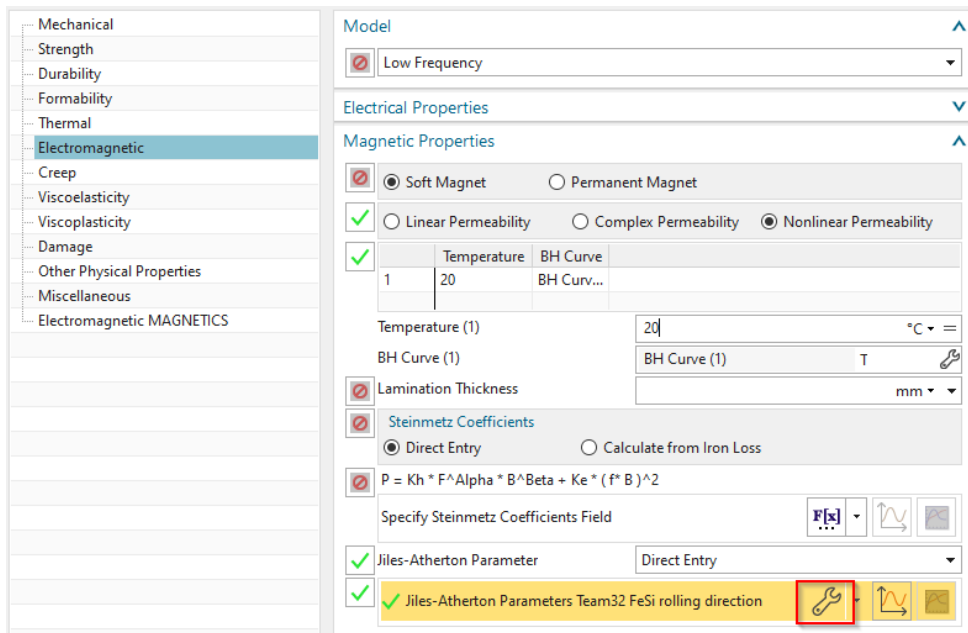
1. In Simcenter, open the Sim file 'Team32\_sim1.sim'.  
Set the displayed part to the Fem file.



2. Check the existing meshes and physical properties. Notice also, there are 1-D circuit elements included to connect the coils in series and apply additional resistances. Two remaining coil connectors are connected with the voltage load as seen later in the Sim file.



3. To model magnetic hysteresis by Jiles-Atherton the material description must have Jiles-Atherton-Parameters. We check this for the core material: Edit the core material (t32\_Core\_mat\_hyst) and open register 'Electromagnetic'.



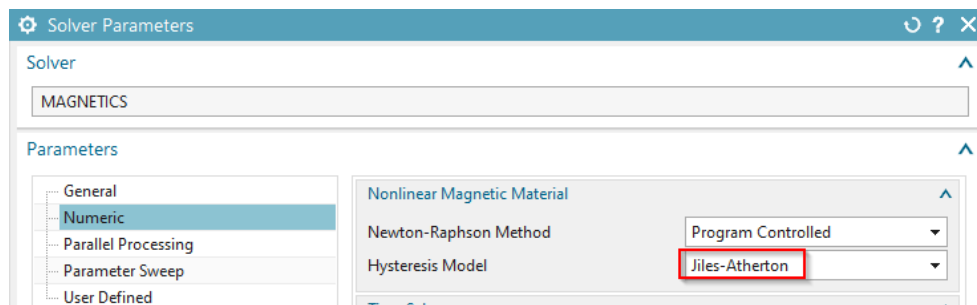
4. At the very bottom of the menu, there are the definitions for 'Jiles-Atherton Parameter'. Notice, these parameters are accessible only if in register 'Electromagnetic' the 'Model'

is set to 'Low Frequency' and if the 'Magnetic Properties' are set to 'Soft Magnet' and 'Nonlinear Permeability'.

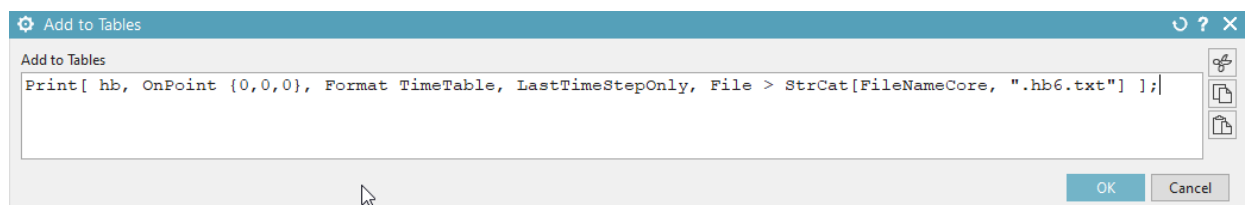
- Click the 'Edit' button (at very bottom of the menu) to open the field editor and display the five parameters. These parameters now will describe the nonlinear and hysteretic material behaviour. The bh curve will not have an influence anymore if Jiles-Atherton is active.

Page Control						
F	t.	Saturation Magnetization (Ms) (A/m) *	Inter-domain Coupling Coefficient (alpha) *	Domain Wall Density (a) (A/m) *	Domain Wall Flexibility Coefficient (c) *	Pinning Coefficient (k) (A/m)
1	20	1330000	0.000417	172.856	0.652	232.652

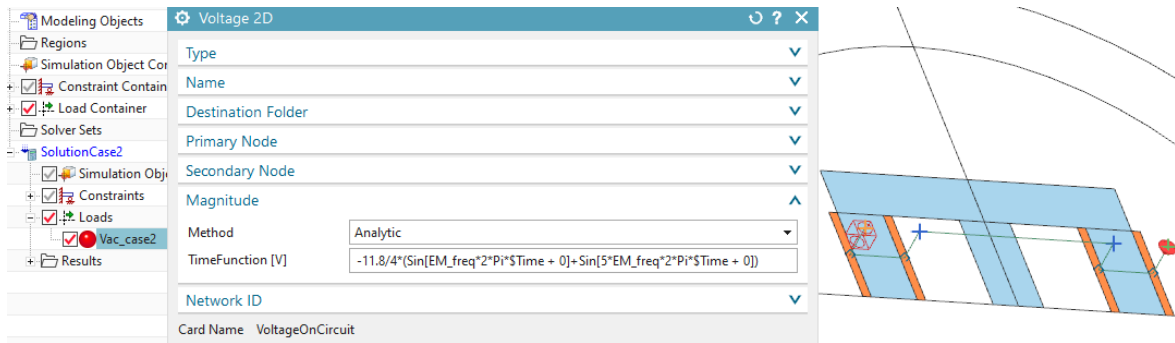
- Close the material menu and set the displayed part to the Sim file.
- Edit the solver parameters of the solution. Switch to register 'Numeric'. Notice, the 'Hysteresis Model' is set to 'Jiles-Atherton'. Thus, the solution will include additional terms for Jiles-Atherton calculation.



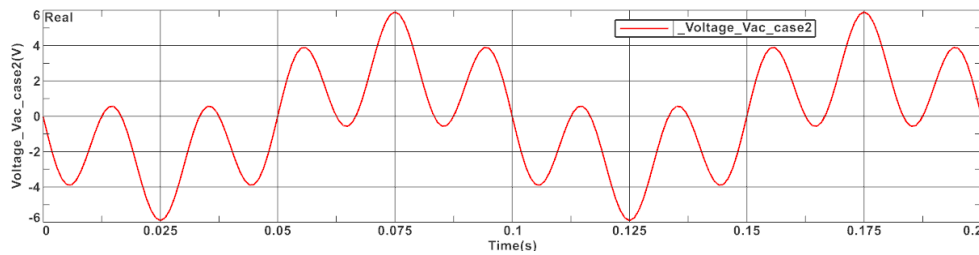
- Be aware that these terms are of nonlinear type and will lead to higher computation time. Thus, also the settings at 'Newton-Raphson Method' do have an influence on this computation. In our case all settings here are at their defaults but other cases may need extra attention here.
- Set the register to 'User Defined'. Notice, in 'PostOperation', 'Add to Tables' there is a text entry defined. This leads to writing the result 'hb' into a text file with extension 'hb6.txt' (6 for position 6). The 'hb' quantity is only for post processing. It contains the x, y, z components of magnetic field strength (h) and x, y, z components of magnetic flux density (b). We will later use this output to generate the hysteresis curve in MS Excel.



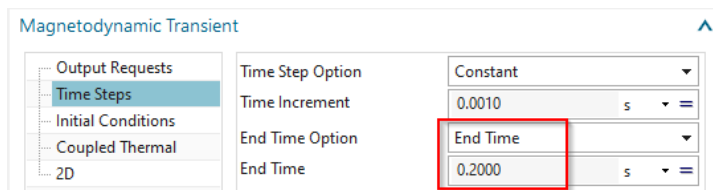
- Check the load: Notice, it is a voltage load being applied to the two coil connectors. The load is defined as 'Analytic', because this allows to add the fifth harmonic simply by writing the desired formula as seen in the below picture.



The following picture shows that voltage as a graph.



11. Edit the solution, in 'Time Steps', set the 'End Time Option' from 'Number Time Steps' to 'End Time'. 'End Time' is then 0.2 sec. Now the solution will run two times over the sinus period.

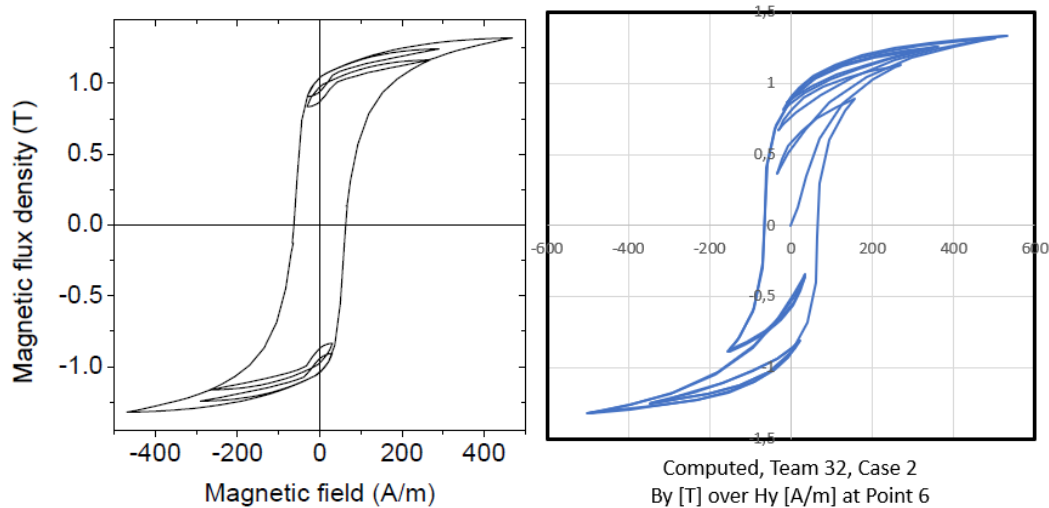


12. Finally, solve the solution. Solve time will be about 3 minutes for the 200 time steps and nonlinear iterations.
13. To make a graph b over h and demonstrate the hysteresis effect, do the following:

- Import the text file ...hb6.txt in Excel (maybe it is necessary to replace points by comma first in a text editor).
- The file contains in each line the result of one time step. There are 3 components for h (magnetic field) and 3 for b (magnetic flux density). Additionally there is the point location and time at the beginning of the line. We now want to make a graph h over b with the y components.

	A	B	C	D	E	F	G	H	I	J	K	L
1	Column1	Column2	Column3	Column4	Column5	Column6	Column7	Column8	Column9	Column10	Column11	Column12
2	0	0	0	0	0	0	0	0	0	0	0	0
3	0	0,001	0	0	0	1,86532E-06	15,34796835	0	1,50934E-08	0,125943664	0	0
4	0	0,002	0	0	0	3,2829E-06	36,49520815	0	3,29276E-08	0,346803435	0	0
5	0	0,003	0	0	0	-1,48912E-05	69,80549716	0	-7,7576E-08	0,615813871	0	0
6	0	0,004	0	0	0	-1,78664E-06	127,8043905	0	-3,2179E-08	0,862141189	0	0
7	0	0,005	0	0	0	6,17895E-06	200,8491664	0	1,48427E-09	1,029204918	0	0

- In Excel, select the two columns and insert an X Y (Scatter) graph. It shows the desired magnetic hysteresis curve. The computed curve (below right) agrees quite well to the reference one (below left).



- The tutorial is complete.

#### References:

- Description of TEAM Problem: 32. "A Test-Case for Validation of Magnetic Field Analysis with Vector Hysteresis", at the following Link.  
<https://www.compumag.org/wp/wp-content/uploads/2018/06/problem32.pdf>
- Experimental data to the TEAM 32 benchmark is available at the following link.  
<http://www.cadema.polito.it/team32>
- "Using a vector Jiles-Atherton hysteresis model for isotropic magnetic materials with the FEM, Newton-Raphson method and relaxation procedure", Guérin C., Jacques K., Sabariego R. V., Dular P., Geuzaine C. and Gyselinck J.
- "Incorporation of a Jiles-Atherton vector hysteresis model in 2D FE magnetic field computations, Application of the Newton-Raphson method", Gyselinck J., Dular P., Sadowski N., Leite J., Bastos J.P.A.

## 9.6 Post-only - Run separate Post-Processing

Usually, when a solve is performed, the system first does a solve and then automatically adds a post-processing of the user-requested outputs (results). The solve may be very time consuming while the post-processing normally needs much less time. A disadvantage is the following: If, after such a solve, the user realizes that another result type may be desired, the whole solve and post-processing work must be done again. To overcome this there exists the feature 'Post-only'. It allows to perform post-processing alone. Of course, there must be a solve that already has run before this. The following steps show such process.

- Open a Sim file and solve a solution.
- Edit the solution and change the 'Output Requests'. Activate any new output and deactivate the existing ones. (Deactivation should be done because otherwise these old results will be written into the result file once again even if they are there already.)
- Edit the 'Solver Parameter' of the solution and in register 'General', activate the bottom 'Solve-only'.

- Check that the bottom 'SaveSolution (for later Post-only)' is activated.
- Solve the solution again. Now, only the post-processing step is done and the newly requested outputs are added into the result file.

The following known issues exist with this feature:

- When the solution uses time steps, the added results will be one step too early. For instance the result of step 5 will be placed in the result file as step 4.
- Any results of type 'Virtual Force' (or moment) cannot be used with the 'Post-only' feature. The reason is that those results need separate solves to be done and if this is not active in the main solve it cannot be post-processed.

This feature is available since NX version 2306. In older versions, it is available if the Magnetics-installation is done with option 'Use Typical Installation': 'N' (no) and 'Latest Version'.

## 10 Error Diagnostics

Some common error messages, usual problems and possible solutions are described in this chapter. A link to the mumps users guide is provided as follows. [http://mumps.enseeiht.fr/doc/userguide\\_5.2.1.p](http://mumps.enseeiht.fr/doc/userguide_5.2.1.p)

### 10.1 MUMPS Equation Solver Messages

In many cases problems result from the equation solver. So, if the solution does not give any results, or all results are zero or infinity, then the feature 'Additional Logfile Output' should be activated. This is done in the 'Solver Parameters' (RMB on the solution) in register 'Numerical'. To make the button visible, first the 'Equation Solver' must be set to 'MUMPS, direct (Default)'. With this feature activated the solver should be started again and there will be a large section of additional messages in the solver logfile. At the end of these messages check for INFOG(1) and INFOG(2). If these show negative values, this means an error occurred. Following is a list of usual messages, explanations and some workarounds:

- INFOG(1) = -1  
An error occurred on processor INFO(2).
- INFOG(1) = -2  
NZ is out of range. INFO(2)=NZ.
- INFOG(1) = -3  
MUMPS was called with an invalid value for JOB. This may happen for example if the analysis (JOB=1) was not performed before the factorization (JOB=2), or the factorization was not performed before the solve (JOB=3). This error also occurs if JOB does not contain the same value on all processes on entry to MUMPS.
- INFOG(1) = -4  
Error in user-provided permutation array PERM\_IN in position INFO(2). This error occurs on the host only.
- INFOG(1) = -5  
Problem of REAL workspace allocation of size INFO(2) during analysis.
- INFOG(1) = -6  
Matrix is singular in structure.
- INFOG(1) = -7  
Problem of INTEGER workspace allocation of size INFO(2) during analysis.
- INFOG(1) = -8  
MAXIS too small for factorization. This may happen, for example, if numerical pivoting leads to significantly more fill-in than was predicted by the analysis. The user should increase the value of ICNTL(14) or the value of MAXIS before entering the factorization (JOB=2).
- INFOG(1) = -9  
The main internal work array S is too small. Increasing parameter ICNTL(14) may help. Try values between 20 and 100. Also, in many cases this results from missing material

properties. Especially this happens if there is zero electric conductivity applied to an electric body.

- INFOG(1) = -10

Numerically singular matrix. The system is somehow under determined and cannot be solved. In many cases this results from missing material properties. Especially this happens if there is zero electric conductivity applied to an electric body. But also stranded coils with missing loads will lead to this problem.

- INFOG(1) = -11

MAXS too small for solution. See error INFO(1)=-9.

- INFOG(1) = -12

MAXS too small for iterative refinement. See error INFO(1)=-9.

- INFOG(1) = -13

The computer didn't have enough memory available. Activating the Out-Of-Core feature (if not already done) can maybe overcome this. Reducing the number of elements or using more memory are other approaches. [6pt]If -13 appears, the requested working memory is larger than the available memory. Working memory is either estimated by the system or can be defined by parameter ICNTL(23).

- INFOG(1) -14

MAXIS too small for solution. See error INFO(1)=-8.

- INFOG(1) -15

MAXIS too small for iterative refinement and/or error analysis. See error INFO(1)=-8.

- INFOG(1) -16

N is out of range. INFO(2)=N.

- INFOG(1) -17

The internal send buffer that was allocated dynamically by MUMPS on the processor is too small. The user should increase the value of ICNTL(14) before entering the analysis (JOB=1).

- INFOG(1) -18

MAXIS too small to process root node. See error INFO(1)=-8.

- INFOG(1) = -19

The maximum allowed size of working memory is too small to solve this model (in the factorization phase). Working memory is either estimated by the system or can be defined by parameter ICNTL(23).

- INFOG(1) = -21

Incompatible values of PAR=0 and NPROCS=1. INFO(2)=NPROCS. Running MUMPS in host-node mode (the host is not a slave processor itself) requires at least two processors. The user should either set PAR to 1 or increase the number of processors.

- INFOG(1) = -22  
A pointer array is provided by the user that is not associated or that has insufficient size. INFO(2) points to the pointer array having the wrong format.
- INFOG(1) = -23  
MPI was not initialized by the user prior to a call to MUMPS with JOB=-1.
- INFOG(1) = -90  
Error in out-of-core management. Check the following: Does the directory for out of core files exist? Is there enough disk space?
- INFOG(2) = -2697 (or any other negative value). [6pt]The system was trying to allocate an extra  $2697 \times 10^6 = 2.697 \times 10^9$  units of memory. If you are using 64 reals this would be  $2.697 \times 10^9 \times 8$  bytes, or 21.6 GB. If you are using 64 complex numbers it would be twice this. This will be the size of an allocate in a single MPI instance. It will also be in addition to any memory already in use.

## 11 Parallel Processing

The Magnetics solver is able to run in parallel. It can use shared memory parallel (SMP), e.g. running with parallel threads on one computer (node) and also distributed memory parallel (DMP), e.g. running spread over several nodes in a cluster. The combination of both SMP and DMP is also possible and very efficient, e.g. running on a cluster while each node uses parallel threads. The DMP feature is possible since version 860. In the following sections we explain how to set up and run that way.

In the following, we will often use the file name of the model to be solved. Therefore we call this 'modelname'. The modelname is the core of the solver input file name. Such solver input files result if in NX or Simcenter the solve option 'Write Solver Input File' is run. File extensions are '.pro', '.msh' and others. If the model for example is the tutorial 'Team24' with the transient solution, then the 'modelname' will be 'Team24-transient'.

### 11.1 SMP on one Computer

If running on one computer, SMP is automatically active.

### 11.2 DMP on one Computer

If running on one computer, DMP can be activated quite simple as follows:

- Right click on a Magnetics solution and chose 'Edit Solver Parameters...'
- In register 'Cluster', at 'Parallel Solution via MPI', activate 'Enable'
- In the field 'Config Parameters', key in 'mpiexec -n 2'.

The value 2 in the above example will activate two parallel processes on this computer. The value can be set as desired, but the recommendation is not to use more that the available number of processors or cores on the computer. Best performance for DMP can be expected if the computer has more than one processor.

### 11.3 DMP on Computer Cluster

To run DMP on a computer cluster proceed as follows.

1. All computers (nodes) must have a Windows operating system. We recommend the version Windows 10 or 11. Possible are also Windows 7 and Windows Server.
2. The nodes must be connected over a network. That network should be of very high performance because of the necessary communication between the processes. If the network is not of high performance, the solve time will be very slow.
3. On each node there must be the same user defined. For this example the user name is 'mpiuser'. The password for this user must be the same on all nodes. Administrator rights are required for that user.

4. On each node there must be a work-directory (wdir) defined at the same path. In this example we use 'C:\workdir\Magnetics'. These directories have to be shared in the network. The mpiuser needs to have read and write permission. Into this directory some files from the MAGNETICS installation folder must be copied. These are

- Magnetics.exe
- MagneticsSolveR.exe
- MagneticsSolveC.exe
- libgcc\_s\_seh-1.dll
- libwinpthread-1.dll
- msmapi.dll

5. On each node the user 'mpiuser' must log in. This can easily be done by remote desktop connection. Using a command shell (CMD) the command 'smpd' must be started with two arguments. The first argument is '-p' and defines the network port through which the solve processes will communicate. Choose a free port there, we will use 19020. The second argument is '-d' and defines the log verbosity. We recommend 3. So, the whole command is this:

- smpd -p 19020 -d 3

Be aware that this smpd process is quite sensitive. In several cases it will hang and must be stopped and restarted. Also take care to use the correct CMD shell. It must be the CMD from the Windows operating system and NOT the NX command prompt.

6. One node will be defined to be the master node. On that master node we will have the solver input files and we will execute the solve command (mpiexec.exe). The master node will spread all child processes to the remaining slave nodes. The master node will have the highest load because several routines that cannot run in parallel will run on the master node.

7. The following solver input files must be copied to the master node work dir:

- modelname.pro (problem definition file)
- modelname.msh (mesh file)
- modelname.Vec.msh (stranded coil directions, only if stranded coils are present)
- modelname.mpi (cluster configuration file, see below)

8. A cluster configuration file (modelname.mpi) must be in the work directory of the master node. The entries are these:

```
mpiexec -hosts NumNodes Node_1 NumProcessesOnNode_1 Node_2 NumProcessesOnNode_2 ... -p NetworkPort -wdir NameOfSharedWorkdir
```

- NumNodes: The total number of nodes in the cluster.
- Node\_1 (Node\_2, ...) The host name of the first (second, ...) node.
- NumProcessesOnNode\_1 (NumProcessesOnNode\_2, ...) The number of processes on the first (second, ...) node.

- NetworkPort: The port number, as already used for the previously defined smpd processes.
- NameOfSharedWorkdir: The name of the shared network directory of the master node.

An example for that modelname.mpi file text content follows. Here, we use the two nodes 'Gaius' and 'Caligula', each running one process while 'Gaius' is the master node:

- mpiexec -hosts 2 Gaius 1 Caligula 1 -p 19020 -wdir \\Gaius\workdir\Magnetics

9. To start the whole parallel solve process the command 'Magnetics.exe', following some arguments must be executed in a command shell on the master node. The arguments are explained below

- Magnetics.exe modelname
- -solve solutionname (the solution name is found in the resolution section of the profile. It can be for instance 'Magnetodynamic\_Frequency3D').
- -bin: for binary write of results. Ascii for readable write.
- Machine1 NumProcessesOn1: The name of the first node following the number of processes to be started on this node.
- Machine2 NumProcessesOn2: Same as above for the next node. And so on.
- -p PortNumber: The network port number for the communication. Use the same as already given with the smpd command.
- -wdir: The name of the shared network folder, that is the same on all nodes.

An example for that CMD command is this:

- Magnetics.exe Team24.sim1-Transient -solve Magnetodynamic\_Transient3D -bin

10. The parallel solve will start. On each node the Windows firewall will probably block the new process 'MagneticsSolveC.exe' or 'MagneticsSolveR.exe'. The execution must then be allowed. On the master node there will be the result files available after finish.

## 11.4 SMP and DMP combined on Computer Cluster

SMP will automatically be active on each node of a cluster solve.

## 12 References

- 'AnderlBinde':  
Anderl, R., Binde, P.: Simulations with NX / Simcenter3D. Kinematics, FEA, CFD, EM and Data Management. Hanser, 2nd Edition.
- 'Geuzaine 2013':  
Geuzaine, C.: ELEC0041. Modeling and Design of Electromagnetic Systems. Université de Liège, Applied & Computational Electromagnetics (ACE)
- 'Kost':  
'Numerische Methoden in der Berechnung elektromagnetischer Felder', Arnulf Kost, Springer-Lehrbuch
- 'Weiland':  
Weiland, T.: A Discretization Method for the Solution of Maxwell's Equations for Six-Component Fields. In: International Journal of Electronics and Communications (AEÜ), Vol.31, No. 3, 1977. p. 116–120
- 'TakahashiNakataMorishige':  
N. Takahashi T. Nakata H. Morishige, (1995), "Summary Of Results for Problem 20 (3-D Static Force Pproblem)", COMPEL - The international journal for computation and mathematics in electrical and electronic engineering, Vol. 14 Iss 2/3 pp. 57 - 75