smartFEM

Analysis and Design of Electrical Drives

User Guide

smartFEM 2.13.07 - 28.09.2023

Table of contents

List of Figures

Introduction

smartFEM is used for simulation electrical machines using alphanumeric respectively graphical representation and for documentation of the calculated characteristics. For this purpose of FEM simulations and analyses, smartFEM uses an FE program called FEMAG, developed by Prof. Dr.-Ing. Konrad Reichert. Numerical techniques are used to solve static electromagnetic field problems by FEM and analytical methods.

The following machine models (motors and generators) can be simulated by using smartFEM:

- Permanent magnet brushless DC motors (internal/external rotor) (BLDC)
- Permanent magnet brush DC motors (internal rotor)
- Permanent magnet linear motors
- Synchronous reluctance motors
- Switched reluctance motors
- Universal motors
- Magnetization of permanent magnet internal rotors

smartFEM was developed using Microsoft® Visual Studio® .NET and it runs on desktop computers with following installed Microsoft® Windows operating systems:

- XP Professional Edition SP 2
- VistaTM Professional
- Windows 7, 8, 10 (32 and 64 Bit)

Delivery:

1 USB-Stick as used as dongle containing the following files:

- xxx.license License to be loaded in smartFEM License Control

- setup.exe Application for starting the installation manually
- smartFEM.msi Installation package (including a FEMAG version suitable for smartFEM version)
- UserGuide_en.pdf User guide
- Topologies_en.pdf Description of rotor and stator topologies

1 Installation

1.1 .NET Framework

Microsoft® .NET Framework 4.0 is required prior to installation of smartFEM. smartFEM issues a verification notice if the required installation is missing. A version can be downloaded from the Microsoft® Download Center and installed.

1.2 smartFEM

Insert the supplied USB stick into the computer, select the "*Start"* button from the Windows start menu and enter the USB drive letters followed by the string *"\:setup.exe"* in the input field. For delivery by email, enter the location of the decompressed file "*setup.exe*"

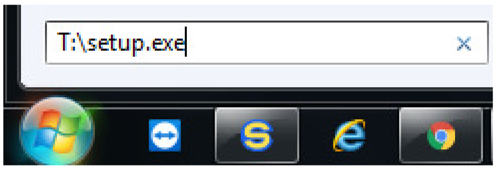


Fig. 1: Window for "Program/File search" - start setup.exe

After pressing the *"Return"* button, the installation program starts and the user will be prompted for all the necessary input data.

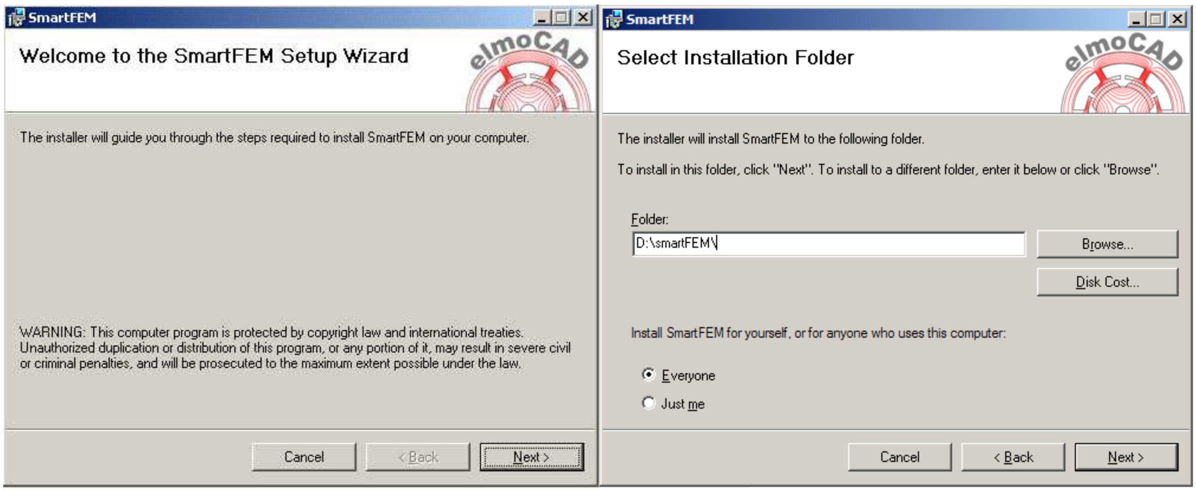


Fig. 2: Installation of smartFEM

Following directories and files are created during the installation process:

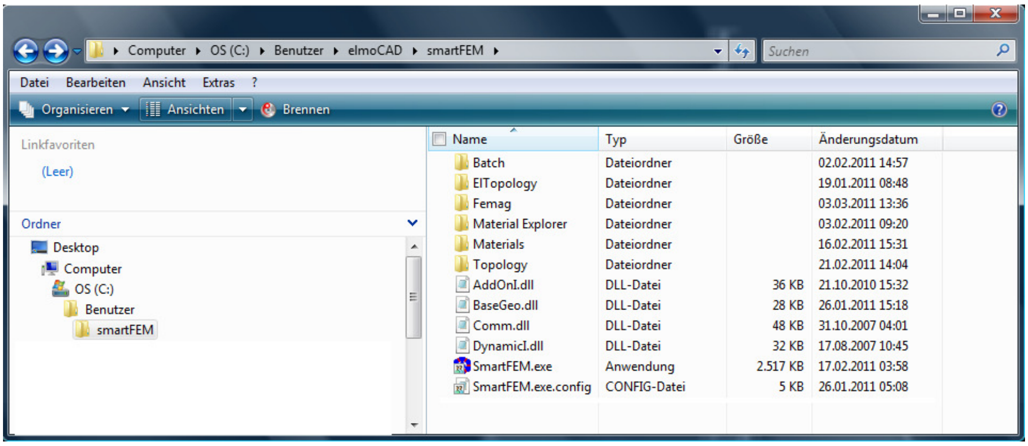


Fig. 3: Program files directory

The directories contain:

... \ smartFEM	the smartFEM.exe program, related DLLs and user guides (*.pdf)
... \ smartFEM \ Batch	templates for performing batch calculations.
... \ smartFEM \ ElTopology	topologies of power electronics for dynamics simulation.
... \ smartFEM \ Material Explorer	program and configuration file for describing and maintaining material data "Material Explorer".
... \ smartFEM \ Materials	material files
... \ smartFEM \ Femag	the program wfemag.exe [1)]
... \ smartFEM \ Topology	dll’s for rotor and stator topologies

[1)] Note: For installation under Windows 7, the directory C:\Program Files\...\smartFEM\Femag must be parameterized with wfemag.exe for writing accesses, as wfemag.exe creates temporary files. If necessary, this directory has to be copied to a user-specific area. smartFEM can also be installed on any other drive and/or folder which can be accessed by user.

The user-specific smartFEM configuration file “user.config” is stored in the current user directory.

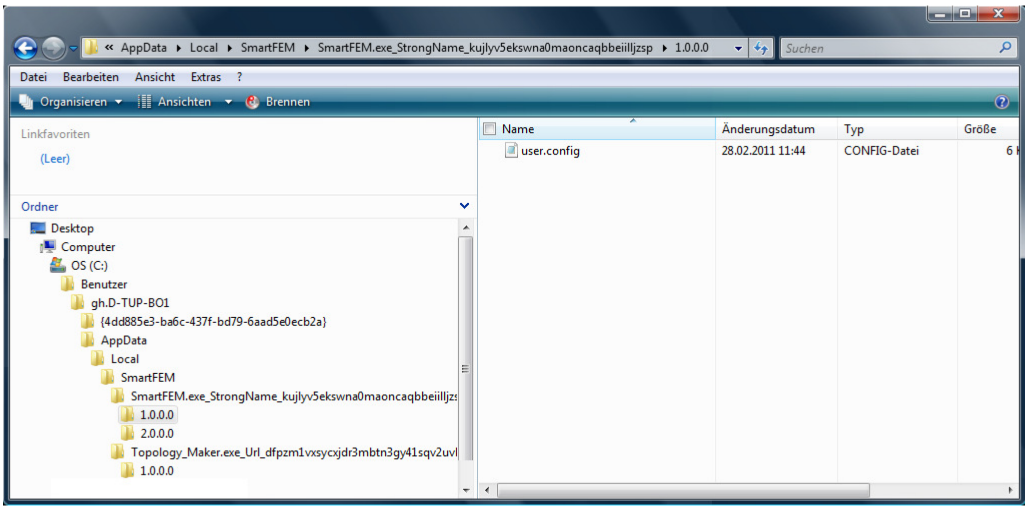


Fig. 4: User-specific files directory

Each time when smartFEM is started, a log file named “smartFEMLogFile.txt” is created and saved in the directory “*C:\Users\User\AppData\Local\Temp\”.* In it is information stored if problems occur.

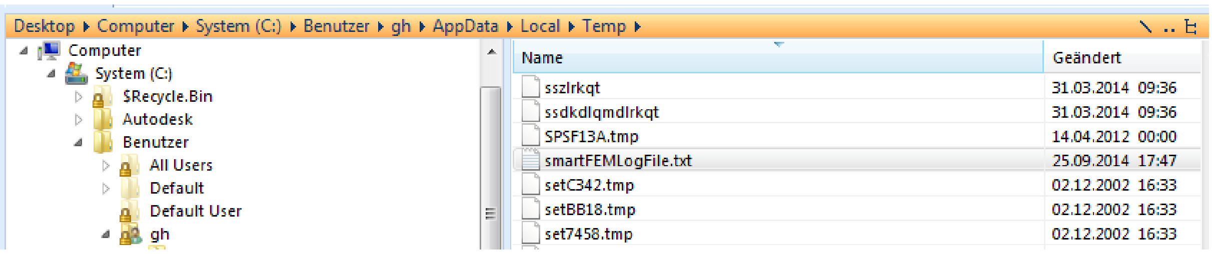


Fig. 5: smartFEM log file

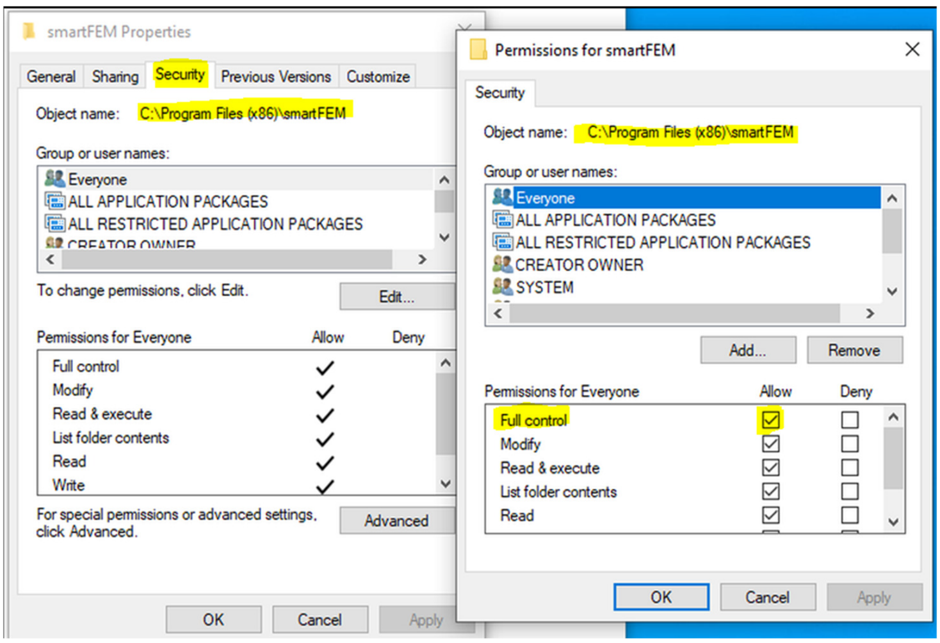
To repair or delete smartFEM, run setup.exe and make the appropriate selection.

Only those files are be deleted which were created during the installation of smartFEM

Fig. 6: Repair or delete smartFEM

Permissions for Windows 10 and 11

If ***smartFEM*** is installed in ***„C:\Programs(x86)\...“***, then has a user with the name ***„Everyone“*** with the permissions ***„Full control“*** be assigned to ***smartFEM***. Otherwise might Windows prohibit the correct use of ***smartFEM***:



2 Start and Settings

2.1 Start

smartFEM is accessed via the Windows *"Start"* menu buttons
"*All Programs"* → *"smartFEM"* → *"smartFEM*".

Fig. 7: Start of smartFEM

A start screen appears. The buttons "*Help"* → *"About ..."* display various information about the installed smartFEM version.

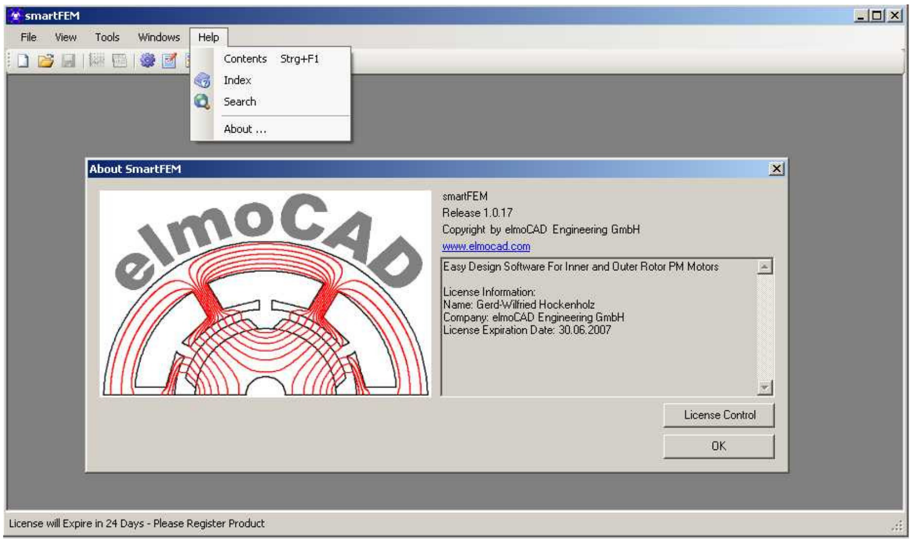


Fig. 8: smartFEM start screen - Information about the installed smartFEM release

2.2 Registration and license information

After the installation, smartFEM is available to the user unrestricted for 30 days. Subsequently no further data storage functions can be performed.

Customers who have purchased a time-limited or unlimited software user license may alternatively use these as follows:

a) Attaching the license to a USB stick. This allows smartFEM being installed on several computers and it can be used after inserting the USB stick on the respective computer.

elmoCAD supplies an USB stick as standard with saved smartFEM software and license files.

b) Attaching the license to the MAC address of the computer, hard disk or network domain as per agreement:

- Create a license file through "*License Control*" → "*Write License Data*"
- Send the generated license file by e-mail to info@elmocad.de
- Read the activated license file activated by elmoCAD and returned by e-mail through *“Help”* → *“About”* → *“License Control”* → *“Load”*

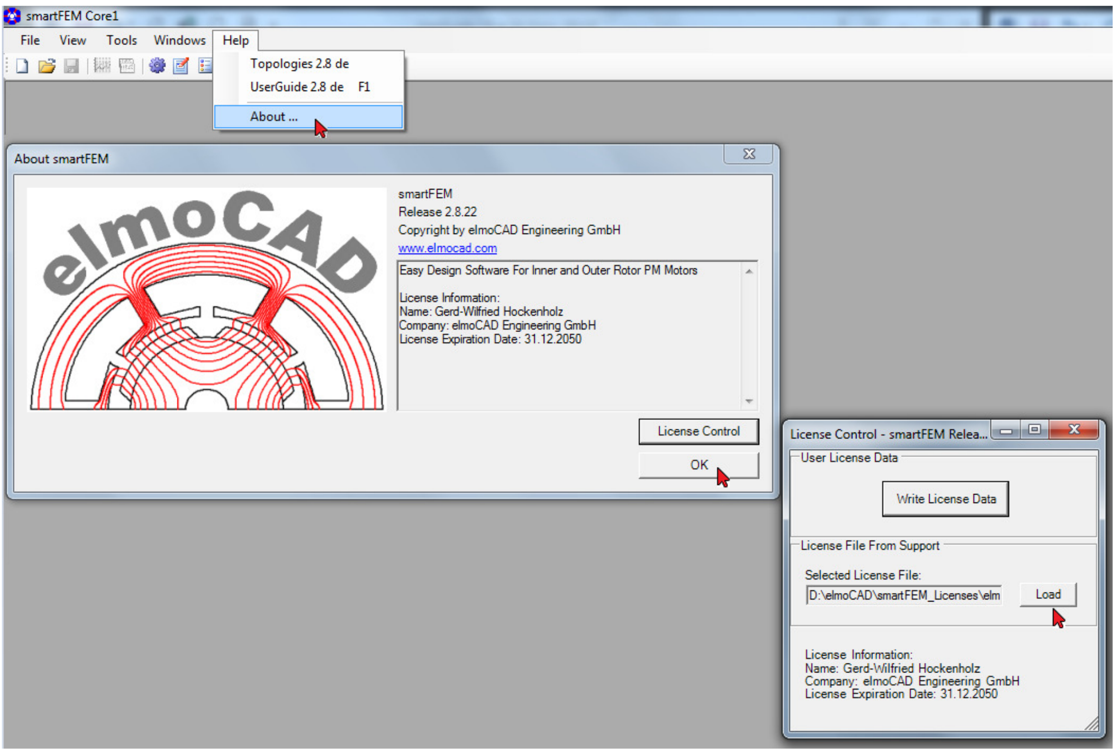


Fig. 9: Loading of the license file

2.3 Settings

2.3.1 smartFEM

Settings for program and data directories, etc. are displayed via the buttons *"Tools"* → *"... Settings"* and can be adjusted.

2.3.1.1 Folders: Settings of directories

- Geometry Topology Folder — DLL files for rotor and stator topologies
- Material Files Folder — files for material data
- FEMAG Exe for smartFEM — The FEMAG version, installed together with smartFEM, by which all smartFEM simulations can be calculated.
- User FEMAG Exe File — a FEMAG version used for additional user-defined calculations. This version can be different from the one used by smartFEM.
- Default Working Directory — Current working directory, where design data and calculation results are stored.
- Electronic Topology Folder — DLL files for electronic circuit topologies

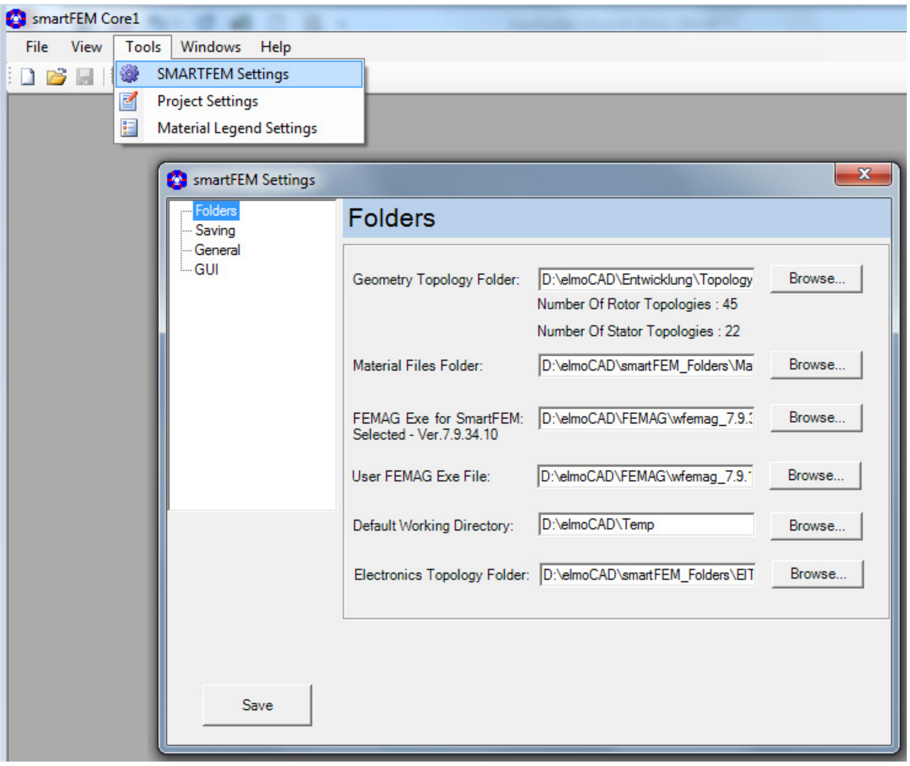


Fig. 10: Tools - smartFEM Settings - Setting the directories

2.3.1.2 Saving: Settings for saving models

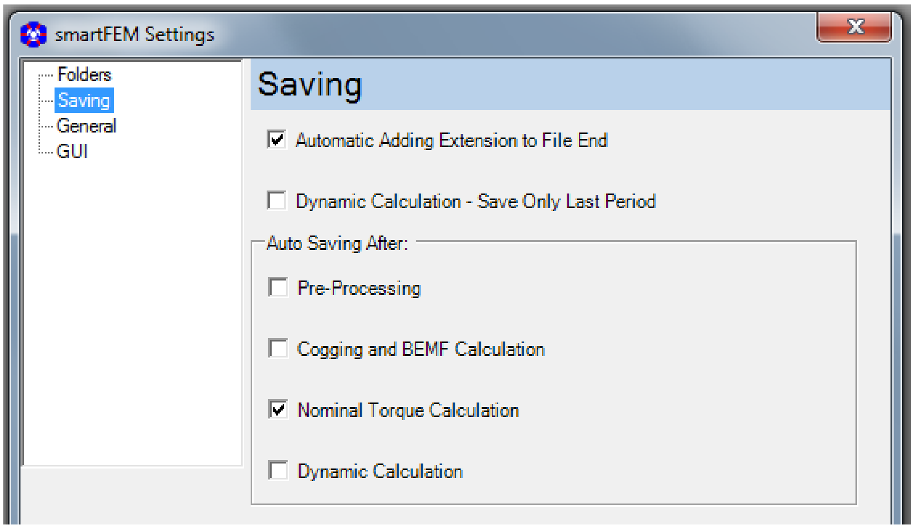


Fig. 11: Tools - smartFEM Settings - Storage of the model data

All model data incl. calculation results are saved in a XMLformated file with name *.mot

- *"Automatic Adding Extension to File End"*
  "_" and an ascending number are attached to the file name for each storage process i.e. filename_1.mot
- *"Dynamic Calculation - Save Only Last Period"*
  Only the result data corresponding the last BEMF period is stored in dynamic calculations.
- *"Auto Saving After"*
  Automatically saves the model data after each selected calculation process:

  - Pre-processing
  - Cogging and BEMF Calculation
  - Nominal Torque and Inductance Calculation
  - Dynamic Calculation

2.3.1.3 General: General Settings

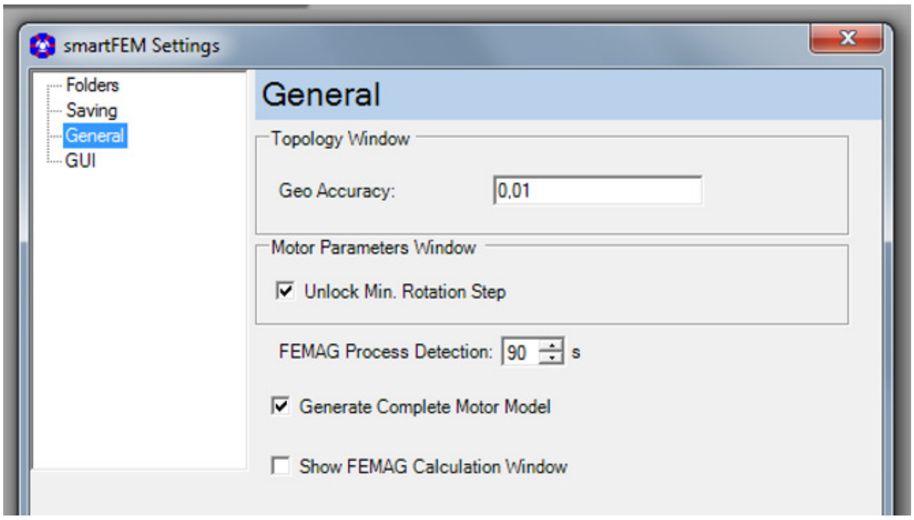


Fig. 12: Tools - smartFEM Settings - General settings

Topology Window:

- *"Geo Accuracy"*
  All geometry points of drawing objects i.e. lines, arcs, etc., which are lying in a circle with defined diameter, are treated as the same point.

Motor Parameter Window:

- Enables entering a user-defined *"Min. Rotation Step "* in window *"Motor Parameters"*.

FEMAG Process Detection:

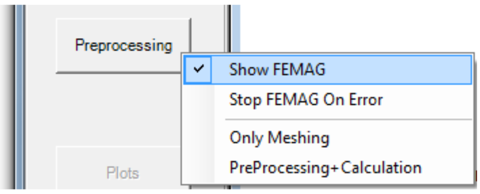
- Enters the monitoring period for a running FEMAG process. If during this time no CPU time is consumed by FEMAG, it is assumed that the calculations performed by FEMAG are either completed and result files are saved or a FEMAG error has occurred and no results are available. In this case, the cause of the error must be clariyfied and the calculation restarted.

Generate Complete Motor Model

- With this is it possible to display in smartFEM main graphic window the complete motor model and rotate it.

Show FEMAG Calculation Window

- With this is it possible to display or hide the FEMAG windows during all FEM calculations. The FEMAG windows can also be switched on and off for single FEM calculations by right mouse click on the related button.



- In FEMAG release 9.7.34.10 windows are generally hidden.

2.3.1.4 GUI: setting for the graphical user interface

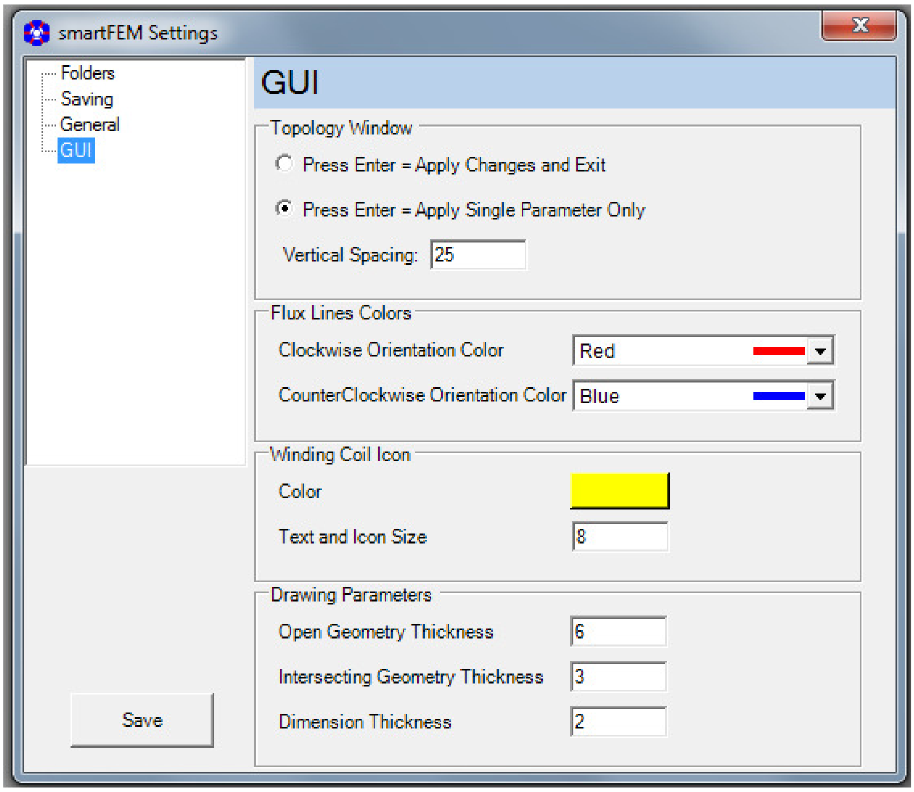


Fig. 13: Tools - smartFEM Settings - GUI settings

Topology Window:

Selection of the action for the *"Return"* key:

- *"Apply Changes and Exit"*
  Apply all changes and exit the topology
- *"Apply Single Parameter Only"*
  Apply the parameter value of the actually selected parameter field whereby the cursor remains in this parameter field.

Flux Lines Colors:

- Definition of the colors of Flux lines used in *"Plots"*. Default color is black.

Winding Coils Icon:

- User defined color and size of icons and text in the graphic window of *"Windings"*.

Drawing Parameters:

- User defined thickness of *"Open Geometry"*, *"Intersecting Geometry"* and *"Dimension"* lines and arcs.

2.3.2 Project

Via *"Project Settings"*, project information such as project number, project name, etc. can be added to a perspective motor design. They are displayed among other things in the project report generated by smartFEM.

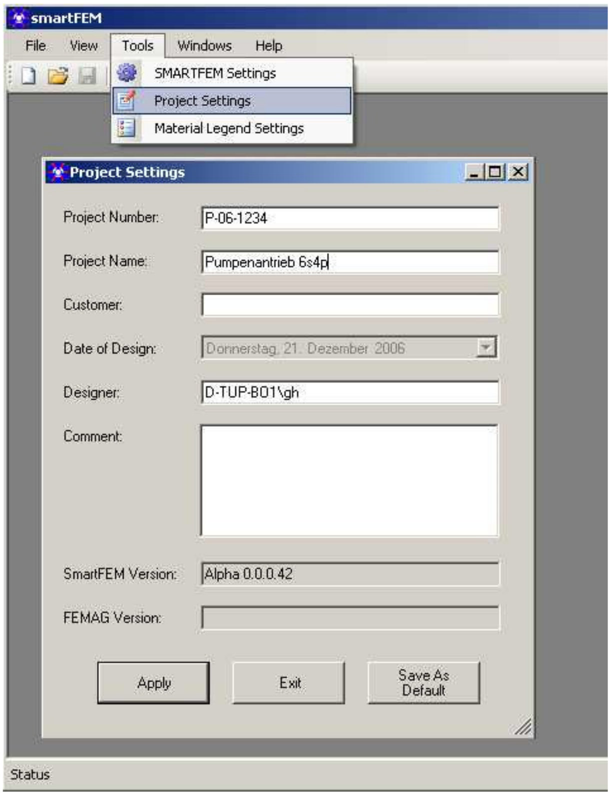


Fig. 14: Tools - Project Settings

2.3.3 Material Legend

Through *"Material Legend Settings"* the colors and designations can be set for surfaces used in the current motor design for each project separately.

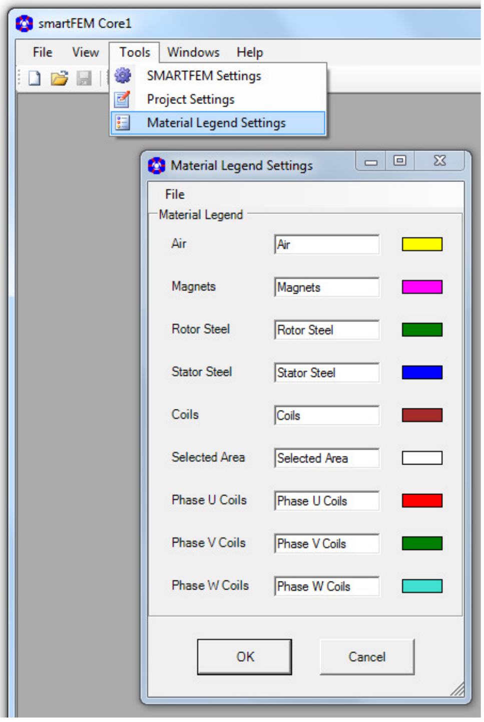


Fig. 15: Tools - Material Legend Settings

3 Developing with smartFEM

1.1 Motor Geometry

After starting smartFEM and selecting the menu *"File → New"*, you can select a motor type. Only those types of motors are displayed for which rotor and stator topologies are stored in the topology list.

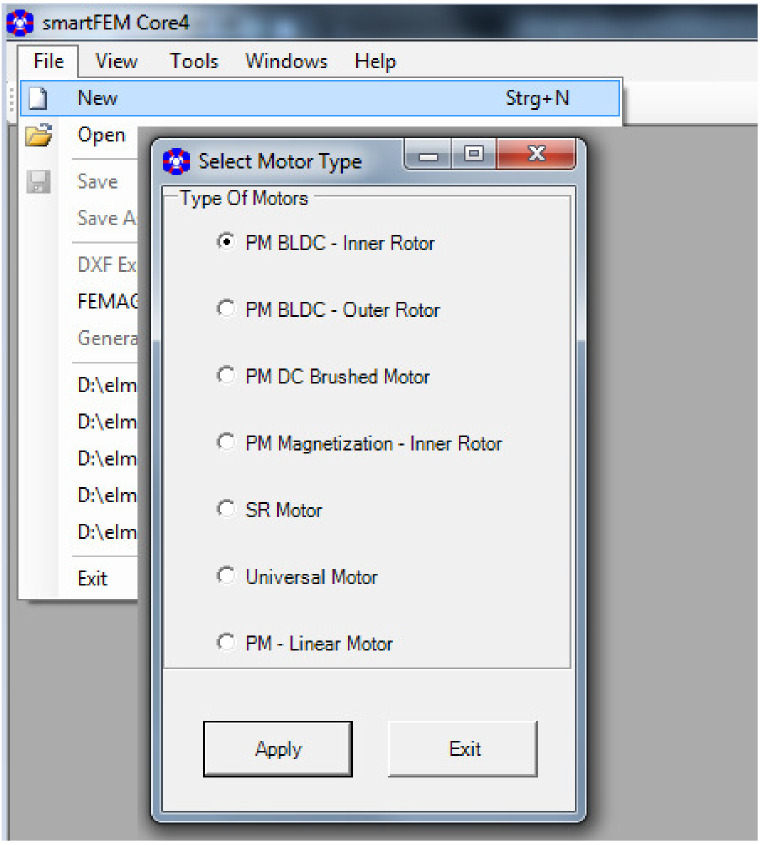


Fig. 16: Creating a new motor type

After pressing the "*Apply*" button, a new motor model is created based on the preset parameters for rotor and stator geometry, and the motor geometry is displayed for a BEMF period in a graph.

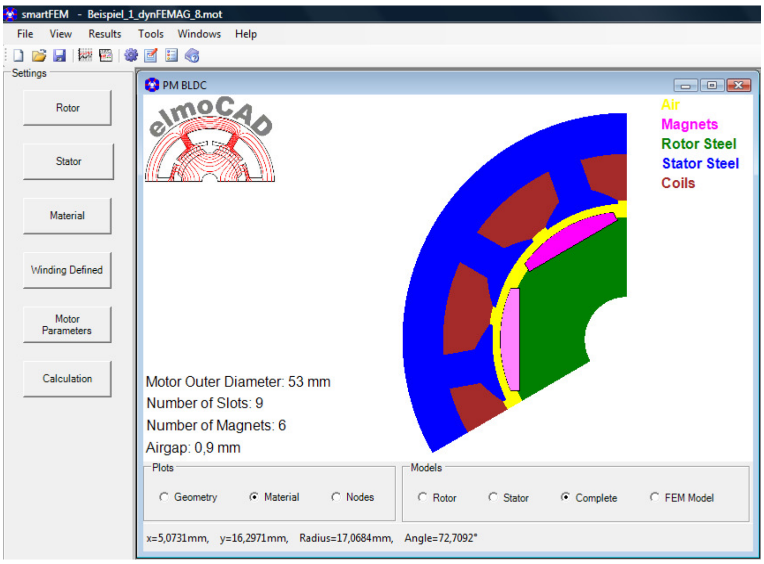


Fig. 17: Motor Geometry

Depending on the selection in the lower area of the geometry window, further geometry views are displayed together with according information contents, like for example:

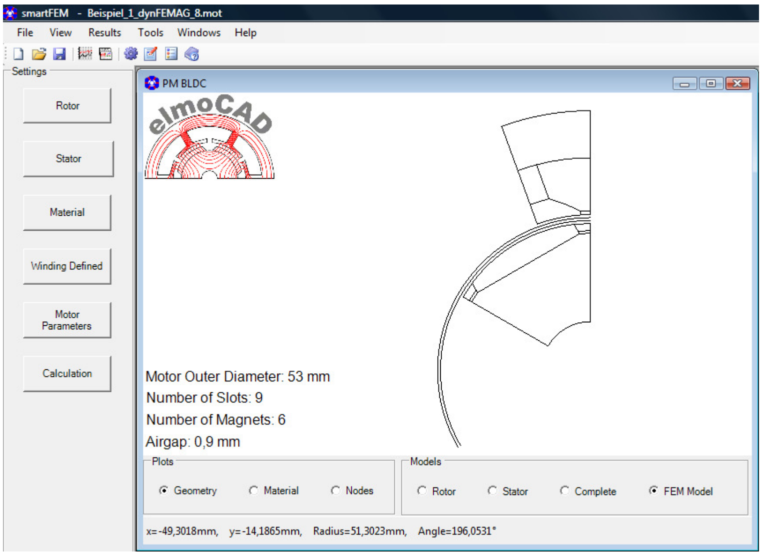


Fig. 18: FEM model with minimal geometrical symmetry of rotor and stator

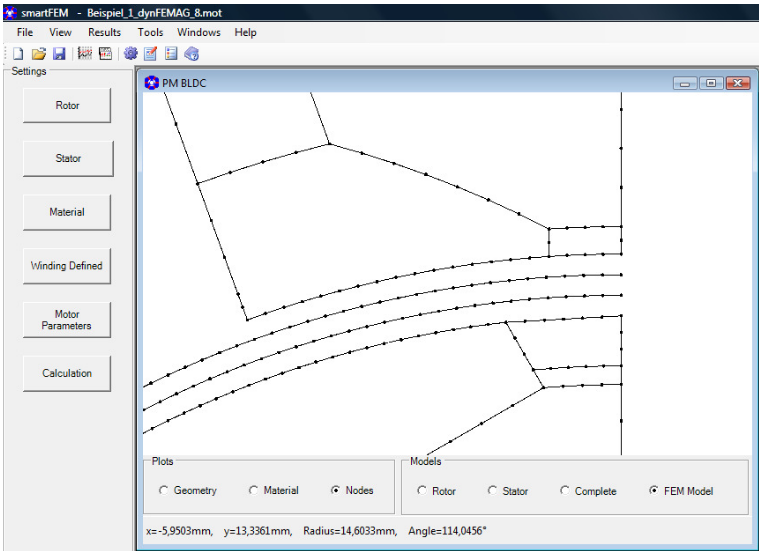


Fig. 19: FEM model zoomed

In all the views, after clicking the left mouse button to a certain position, the scroll wheel can be used to zoom in any content as far as possible.

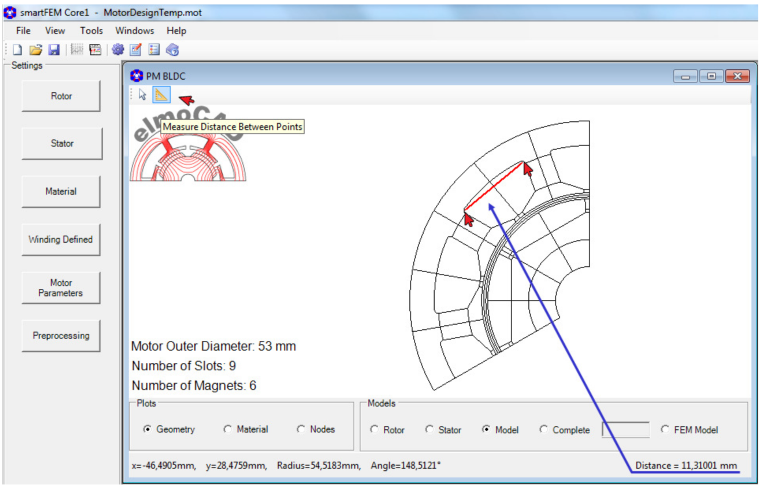


Fig. 20: Measurement of distances

After selecting the button "*Measure Distance Between Points*", distance measurements between two mouse-clicked positions can be performed.

3.1.1 Developing steps

Developing the simulation model up to the first simulation or calculation is best done through the following steps:

- Defining rotor geometry
- Defining stator geometry
  In case of external or internal rotor motors, the internal or external diameter of the stator is automatically adjusted to the external or internal diameter of the rotor.
- Selecting materials
- Input of winding scheme and winding parameters.
- On the base of the selected coil/slot combination of rotor and stator,
  a winding scheme with maximum BEMF is automatically suggested.
- Defining motor and calculation parameters

3.1.2 Rotor

After selecting the "*Rotor*" button, the geometry of the rotor topology is drawn in the graphics window and the corresponding parameters are displayed in a pop-up window in three groups *"Geometry"*, *"Basic"* and *"Elements"*.

By opening the selection field "*Types of Rotor*", all rotor topologies saved in the smartFEM topology folder for the corresponding motor type are displayed.

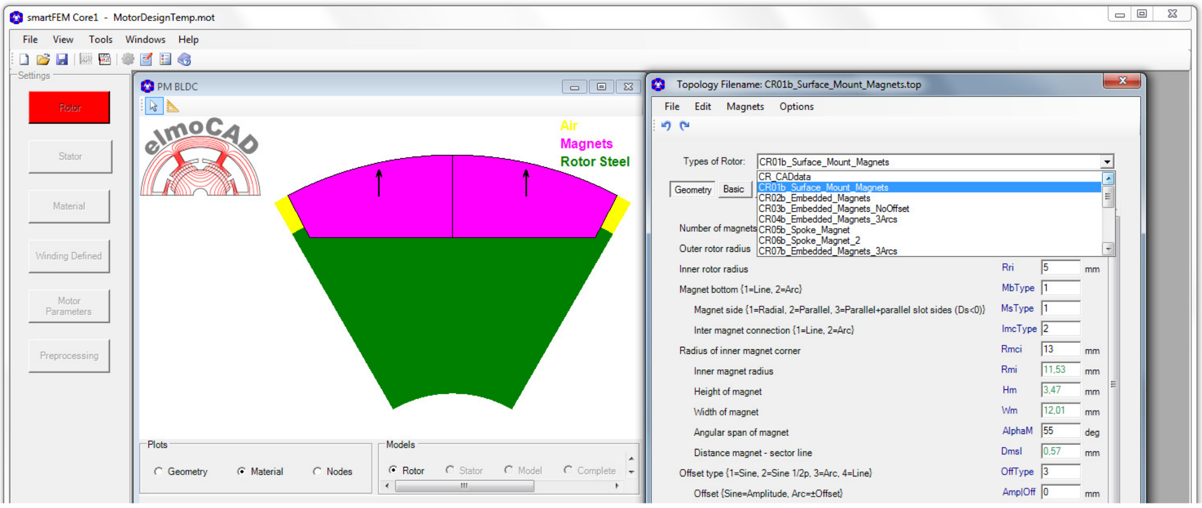


Fig. 21: Selecting a rotor topology

After selection of a topology the geometry parameters are displayed immediately in the graphical window and its associated alphanumerical parameter in the before mentioned three groups (tabs):

- *"Geometry"* these parameter define the geometry based on Cartesian or Polar Coordinates of points which are connected by arcs, lines and/or curves.
- *"Basic"* these parameter describe general values assigned to the geometry
- *"Elements"* these parameter display properties of arcs, lines, curves, areas and points. Different parameter can be edited, i.e. to parametrize node chains.

The geometry can be displayed in the graphical window as:

- drawing of arcs, lines and curves
- colored areas showing the material
- node chains

3.1.2.1 „*Geometry*“ Parameter

The parameters describing geometry are displayed in the parameter group *"Geometry"*. As soon as the cursor is positioned into a parameter field using the mouse or "Tab"-key, parameter specific dimension arrows are displayed as help information in the graphic.

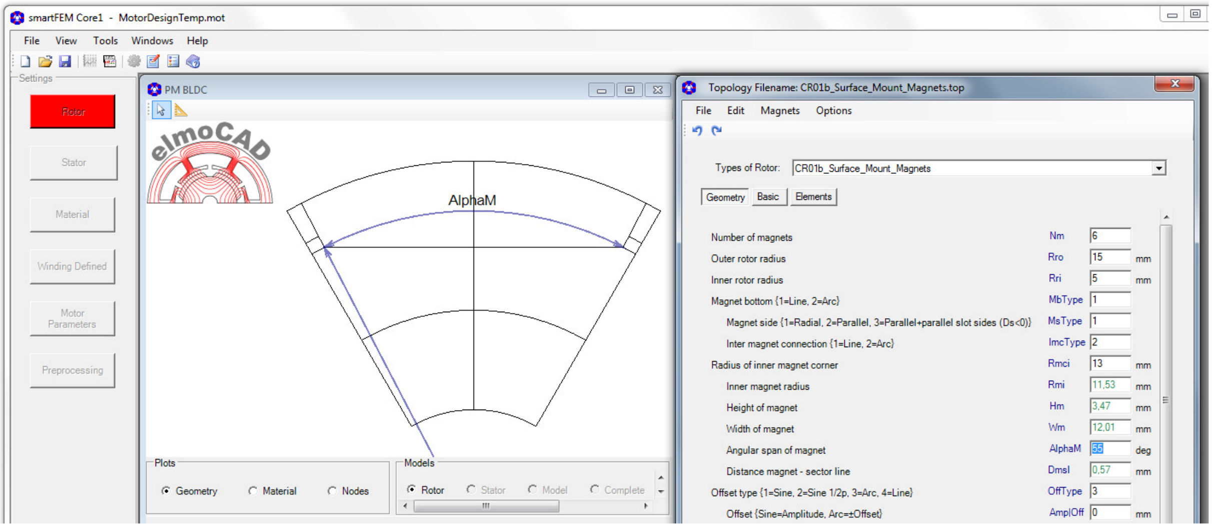


Fig. 22: Rotor - Geometry parameter

Each input of a new value (e.g., Width of Magnet) results in re-calculation of the geometry, which is redrawn directly in the graphic window. Parameter fields with green font contain calculated parameters, which are used for information and cannot be changed.

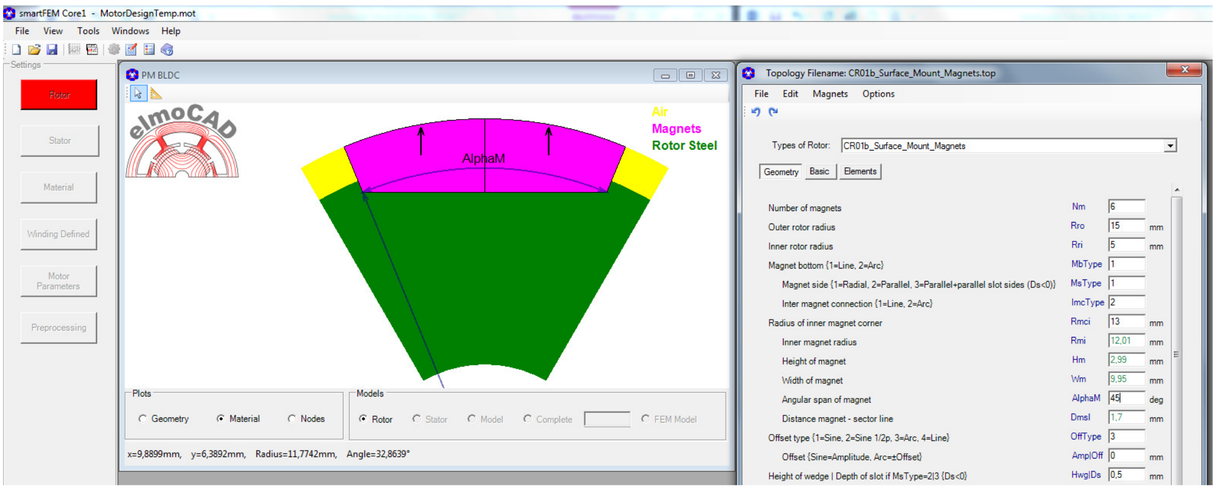


Fig. 23: Rotor - Display of modified geometry

Parameters, which have been changed after selecting the topology, are only accepted and saved after pressing the "*Apply*" button. If you leave the parameter window via the "Exit" button, no changes are accepted and the origin geometry is displayed again.

The “Un-Do” and “Re-Do” buttons allow user to refuse actual input i.e. in case of faulty input and display the previous value.

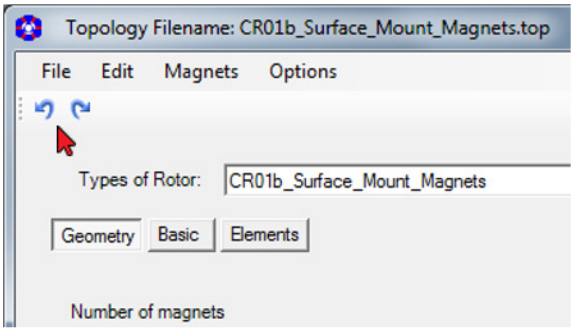


Fig. 24: Rotor - Un-Do and Re-Do

3.1.2.2 „EditGeometry“ – User Defined Geometries

Beginning with smartFEM release 2.11 is possible to edit user defined geometries .

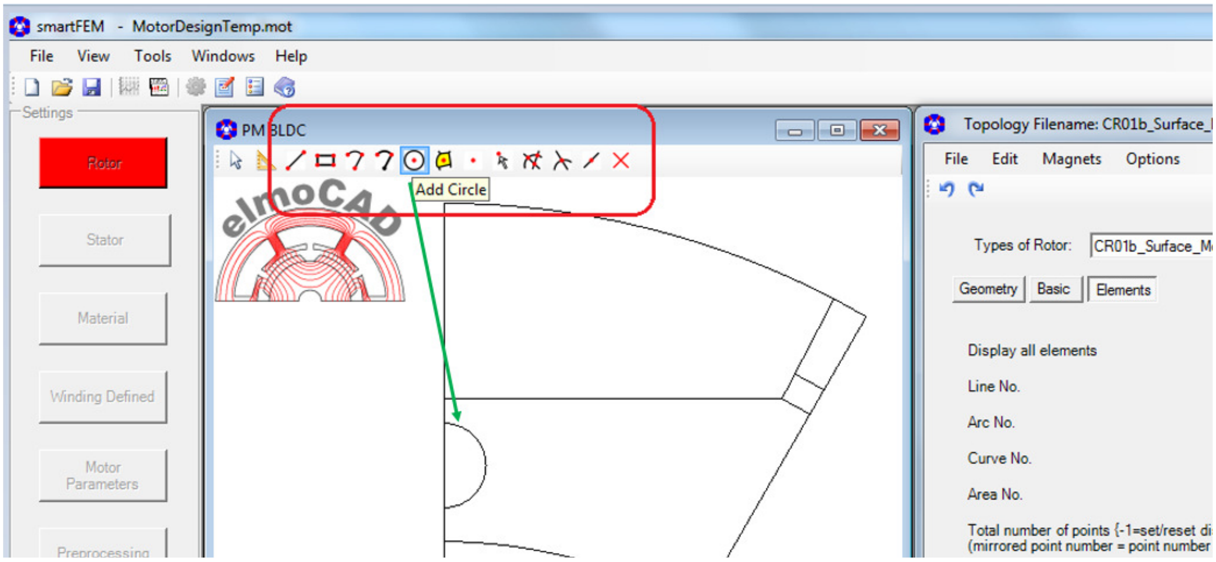


Fig. 25: Edit Geometry - User defined Elements

Details are described in the document ***„Topologies 2.11.00 de.pdf“*** which can also be opened by menu *„Help – About“*.

3.1.2.3 Plausibility Checks

In order to develop geometry, the user can enter parameter values in any order. Due to their dependence on other values, they initially might not appear plausible because they are outside of the currently permissible value range. The Geometry is therefore only consistent when all parameters are entered with correct values.

Immediately after the input of one value, all values are subjected to a consistency check. Depending on the type, the identified incompatibilities are treated as follows:

- If a value is entered which is too small or too large, it can be automatically reset to the permissible minimum or maximum value. I.e. the magnet cannot be wider than the sector of the rotor. The input value of the number of magnets has a higher priority.
- The lines and surfaces affected by the entered value with the resulting overlaps are displayed in the graphic in red color, the *"Apply"* button is renamed to *"Bad Geometry"*

and is also changed into red color. The user has now to correct the last input value or change other values so that the geometry no longer has any inconsistencies.

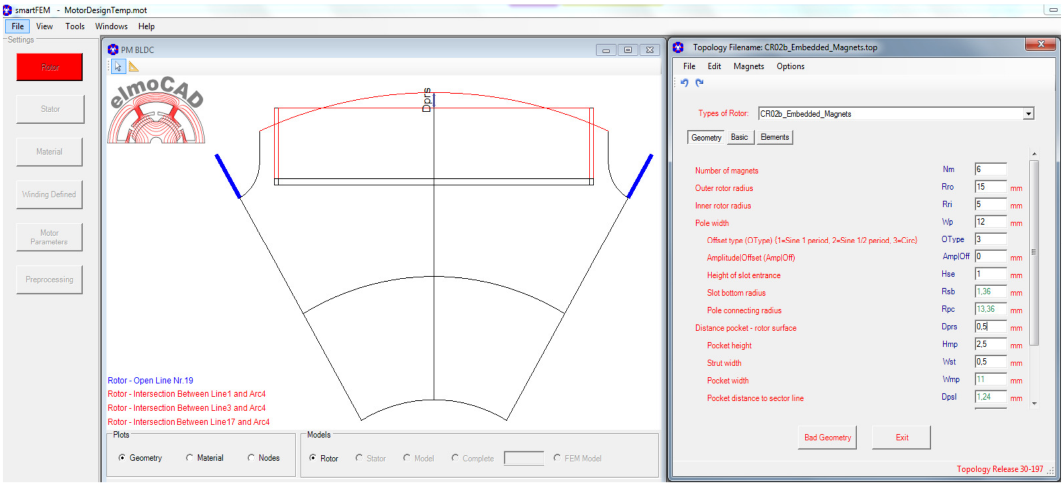


Fig. 26: Rotor - Plausibility checks

- Open lines and arcs are displayed in blue color. It is not a problem if these are positioned on the sector lines to the air gap. These lines are in dependence of the rotor type necessary to connect the rotor geometry to the air gap layer. If further open lines or circular arcs exist after a DXF import of a rotor or stator geometry, this must be checked and corrected in the CAD drawing (see also DFX import). The line thickness can be set in *"smartFEM Settings - GUI"*.

3.1.2.4 „Basic“ Parameter

In the parameter group "*Basic*", the so-called basic parameters can be edited:

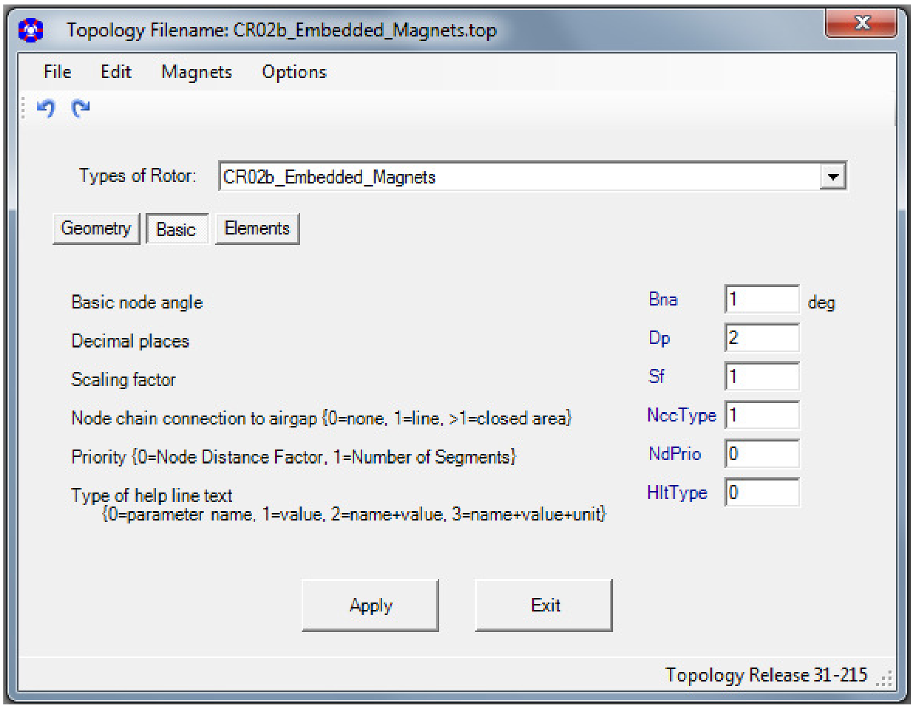


Fig. 27: Basic parameter

- Basic node angle — Minimum node distance at the air gap in deg. A precise definition and explanation is given in 3.1.2.5
- Decimal Places — The number of decimal places displayed.

  All parameters are rounded according to the input or before output.
- Scaling Factor — A scale for enlargement or reduction exclusively for rotor geometry. All input parameters are multiplied by this factor.

  Exception: Internal or external radius of the stator in the case of internal or external rotors. They are calculated from the outside or/and inside radius of the rotor + air gap. In this case, the stator parameters are stretched in the radial direction.
- Node chain connection to air gap — This is used to control the connection of the geometry to the air gap.
- Priority — This is used to control whether the input value *"Node Distance Factor"* or *"Number of Segments"* on the node chains is kept constant when geometry parameter are changed.

3.1.2.5 Node Chains

The base for the required mesh for the FEM calculations are the node chains represented from the drawing elements: *"line"*, *"circular arc"* and *"curve section"*. The node chains are built from every drawing element by segmented lines whereby nodes indicate begin and end of each segment.

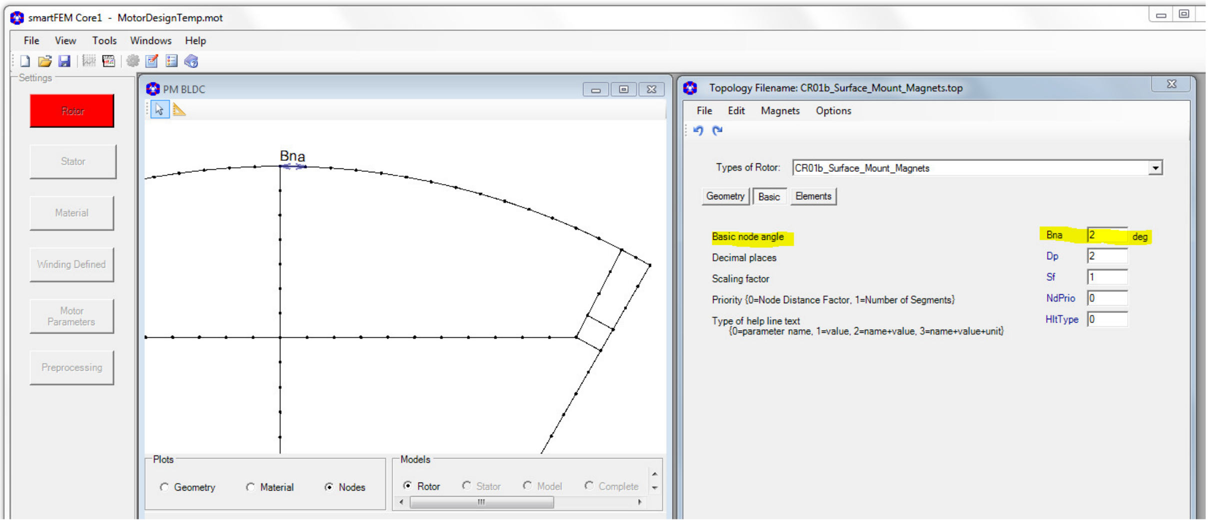


Fig. 28: Node chains with indication of the base node angle

The number of the segments is calculated by:

- Rotating motors:

  Number of Segments = Integer $\left(\frac{\text{element length}}{\text{Bna} \times \text{Node distance factor}}\right)$

  Bna = Base node angle [°mech] respectively [deg]

  = angle between 2 nodes at the air gap arc with center point P(x,y) = (0,0)

  element length in rad]

  Node distance factor = element-specific factor (editable in parameter group "Elements")

- Linear motors:

  Number of Segments = Integer $\left(\frac{\text{element length}}{\text{Bnd} \times \text{Node distance factor}}\right)$

  Bnd = Base node distance [mm]

  = distance between 2 nodes at the airgap line

  Node distance factor = element-specific factor (editable in parameter group "Elements")

Base node angle and element-dependent factors are pre-set for each topology with values that provide a “good” network for the FEM calculations

The user can adjust the base node angle respectively base node distance in the parameter group "Basic". It applies to all drawing elements. The element related factors are adjusted individually in the group "Elements" for each drawing element.

The setting of the node spacing in the central air gap layer, which is important for an accurate power calculation, can be carried out in the dialogue field "Motor Parameters" with the parameter "Min. Rotation Step" (see also 3.6). It should be chosen in such a way, that the network elements get the shape of a square as much as possible.

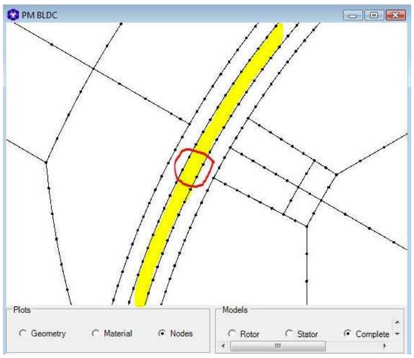


Fig. 29: Node chains in the air gap

3.1.2.6 Adjustment of node chains

By clicking the right mouse button on a drawing element (point, line, arc), the element is highlighted with additional information.

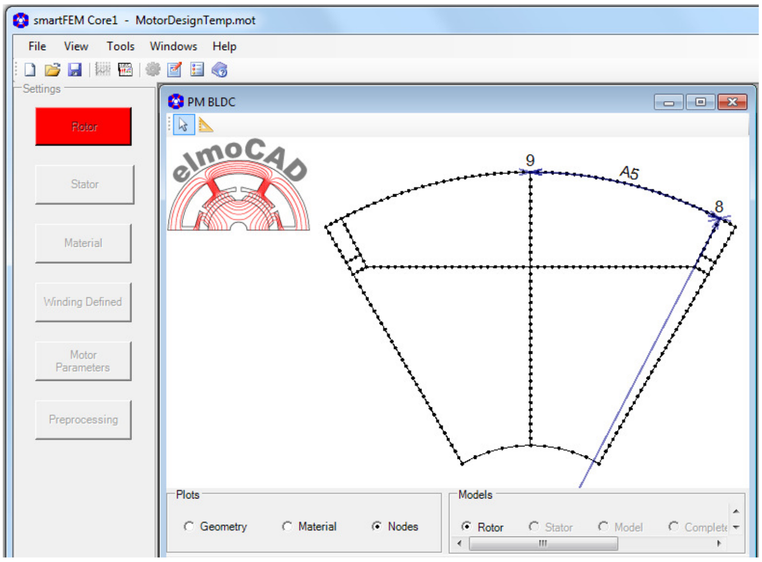


Fig. 30: Rotor geometry - Representation of node chains

Simultaneously, the parameter window changes to the group "*Elements*", the cursor is positioned in the relevant parameter field and additional information is displayed.

An element can also be selected by entering the element number.

The distance between nodes can be entered in the parameter field "*Node distance factor*" as a multiple of the base node distance for the selected element. Subsequently, the number of segments is calculated. When entering the "*Number of Segments*" the "*Node distance factor*" will be re-calculated according to this number.

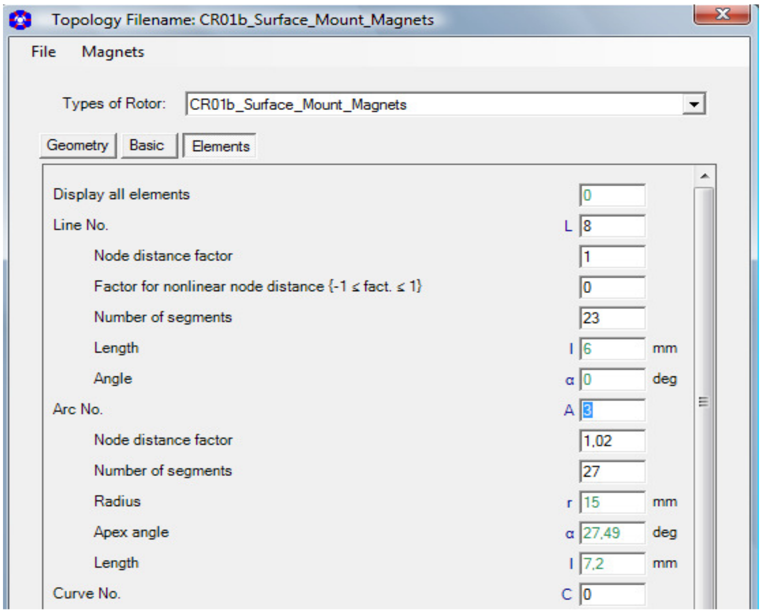


Fig. 31: Rotor geometry - "Elements" group

If the length of the element is changed, the "*Node distance factor*" or "*Number of segments*" is kept constant, depending on the setting of the parameter "*Priority*" in the "*Basic*" parameter group.

When creating lines, non-linear node spacing can be generated with the help of an additional factor -1 ≤ „*Nonlinear node distance factor*“ ≤ 1 in order to adjust distances between nodes at the beginning and end of a line to the adjacent elements. By changing the sign, the direction of the node spacing is reversed.

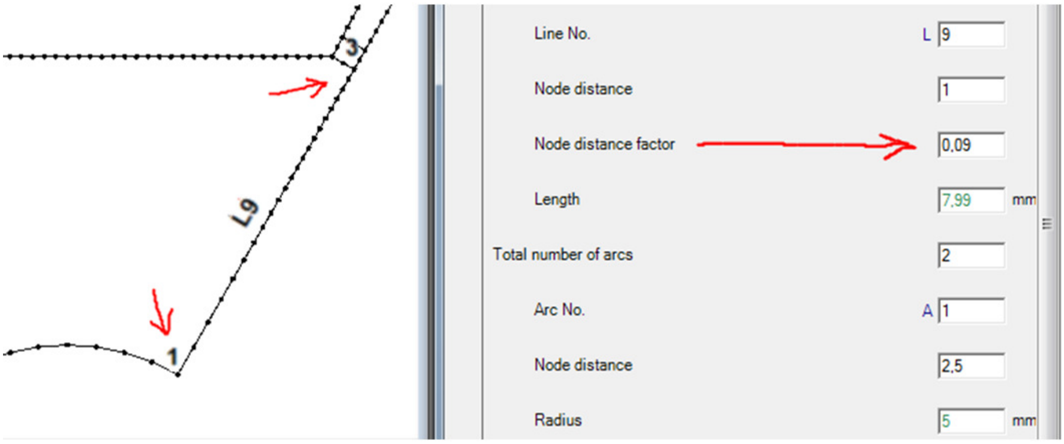


Fig. 32: Rotor geometry - Non-linear node chains

3.1.3 Stator

The stator geometry is, just as the rotor geometry, selected and processed accordingly. Here are also several stator typologies available, which the user can select.

When selecting "*One Coil Per Slot*" or "*Two Coils Per Slot*", either "one-layered" or "two-layered fractional slot winding" will be set as a winding type. In some stator typologies, for "two-layered fractional slot windings", it is possible to select lower/upper layer with equal surface areas, in addition to the position in the left/right slot half.

The editing of node chains is analogous to the rotor.

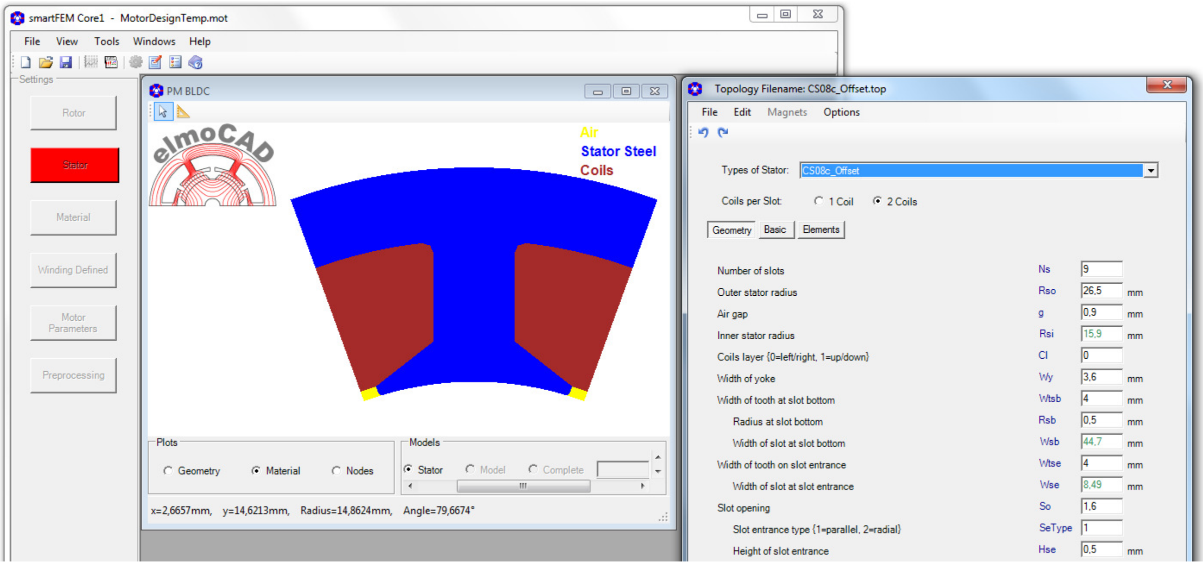


Fig. 33: Stator - Edit

3.1.4 Saving, DXF and FEMAG export

After editing the rotor and stator geometry, the entire model can be saved and the geometry will be exported in FEMAG- and DXF file formats.

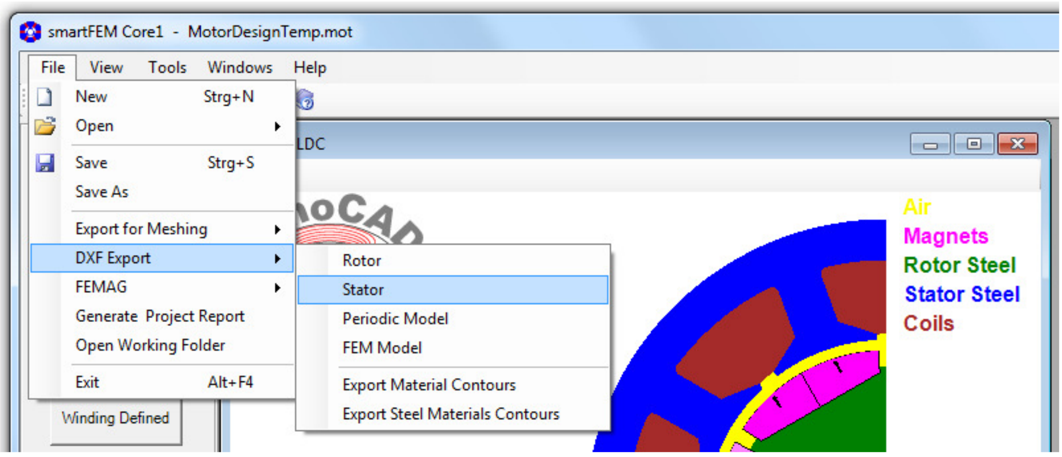


Fig. 34: Saving and exporting a motor geometry

All drawing elements of a topology can be exported into a DXF-File based on the format specification of *"AutoCAD® dxf-reference"*:

- Rotor
- Stator
- Periodic Model
- FEM Model

Additionally can be selected whether only "Material" or "Steel Material" contours should be exported.

In the menu of all topologies can DXF files be created with all the text information regarding the properties of drawing elements, i.e. Basic Parameters and Element Information.

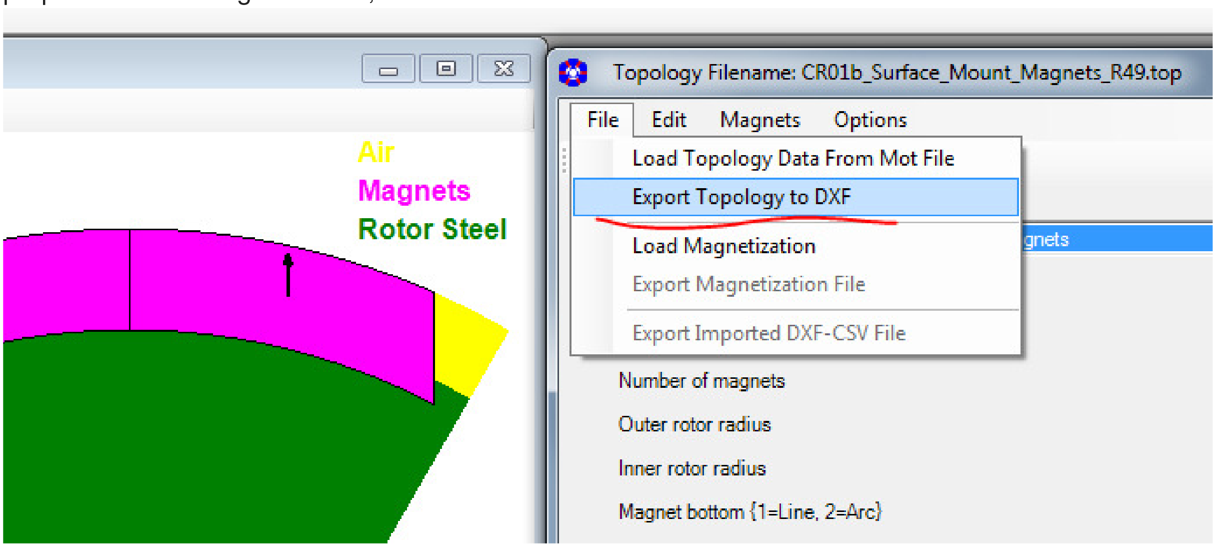


Fig. 35: Saving and exporting a motor geometry

Further details are described in chapter "*7.3 DFX Export"*.

3.2 Materials

By clicking the left mouse button on the "*Material*" button, a pop-up window is opened for the entering of material data for electrical sheets, magnets and coils.

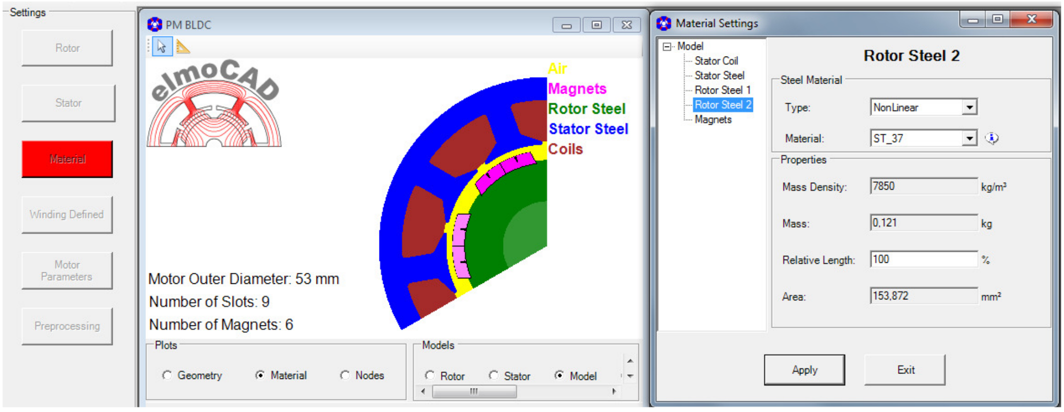


Fig. 36: Material Settings - Assignment of material properties

The *"Relative Length"* of material refers to the length of the motor, which can be edited in *"Motor Parameters"*. It is used together with the material densities for the calculation of masses and moments of inertia and are also taken into account for calculation of BEMF, torque, etc.

3.2.1 Stator Coil

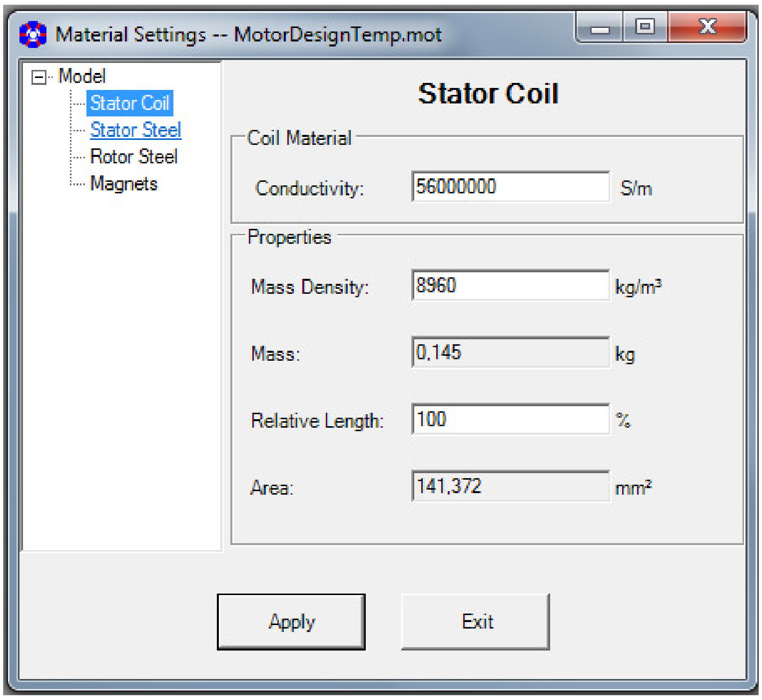


Fig. 37: Material Settings - Stator coil parameter

The parameter values of *"Conductivity"*; *"Mass Density"* and *"Relative Length"* can be edited. Default values are provided for copper material. "Mass" and "Area" of this material of the complete machine are calculated values based on the mass density and related coil of area the stator including head windings.

3.2.2 Stator and Rotor Steel

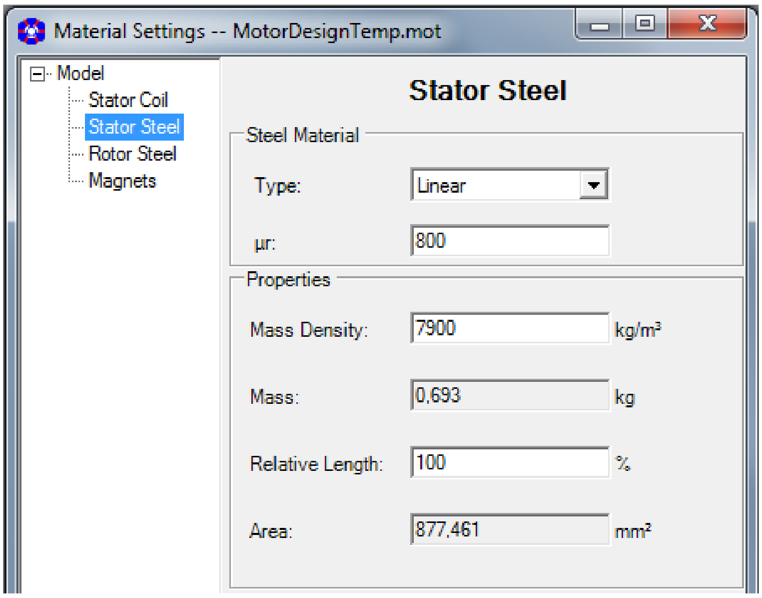


Fig. 38: Material Settings - Stator steel parameter

User can select material type *"Linear"* or *"Non-Linear"* and in case of

- *"Linear"*
  values for *"µr"*, *"Mass Density"* and *"Relative Length"* can be edited.
- *"Non-Linear"*
  A material file containing properties i.e. BH curves, loss data, mass density, etc. can be selected from a drop-down list. Only the *"Relative Length"* can be edited.

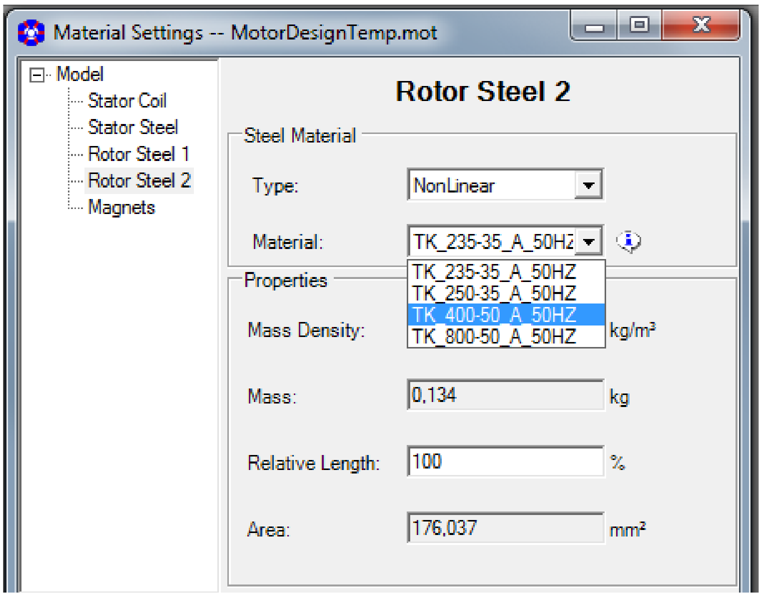


Fig. 39: Material Settings - "Non-Linear" steel parameter

For assignment of different materials in stator and rotor topologies for individual areas can different material numbers *"MatNo"* be assigned in parameter group “Elements” which then are displayed in toned down colors. This has to be done first before the material data can be assinged accordingly to *"MatNo"* in *“Material Settungs”* for stator and rotor steel.

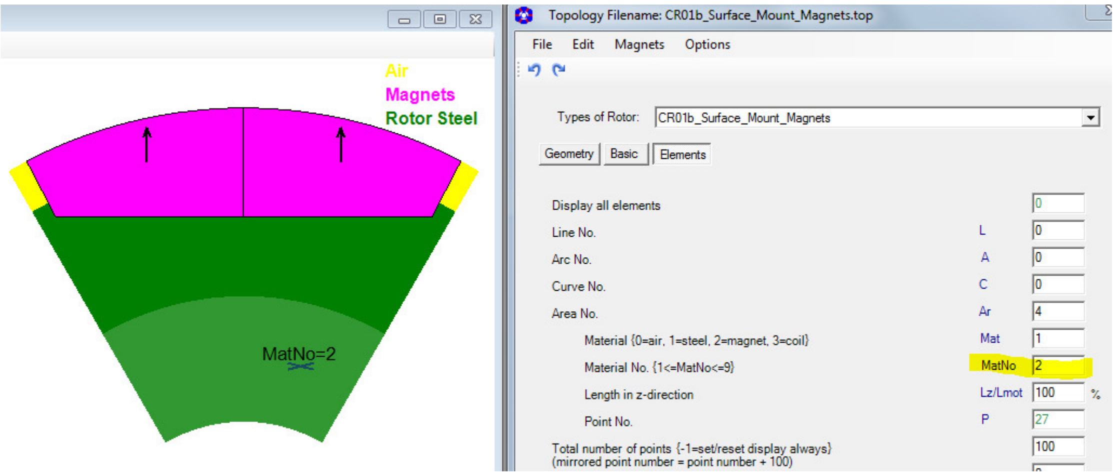


Fig. 40: Material Settings - Assignment of different material numbers “MatNo” for different areas within one topology

Creation and maintenance of the material files with BH-Curves and loss data/coefficients can be done with the *“Material Explorer”* software (see 3.2.5 Material Explorer / Material Editor).

3.2.3 Magnets

For description of the magnet properties in case of “Linear” calculation remanence induction “B_r”, relative permeability “μ_r”, “Conductivity” and “Magnetization” of the magnetic material have to be specified.

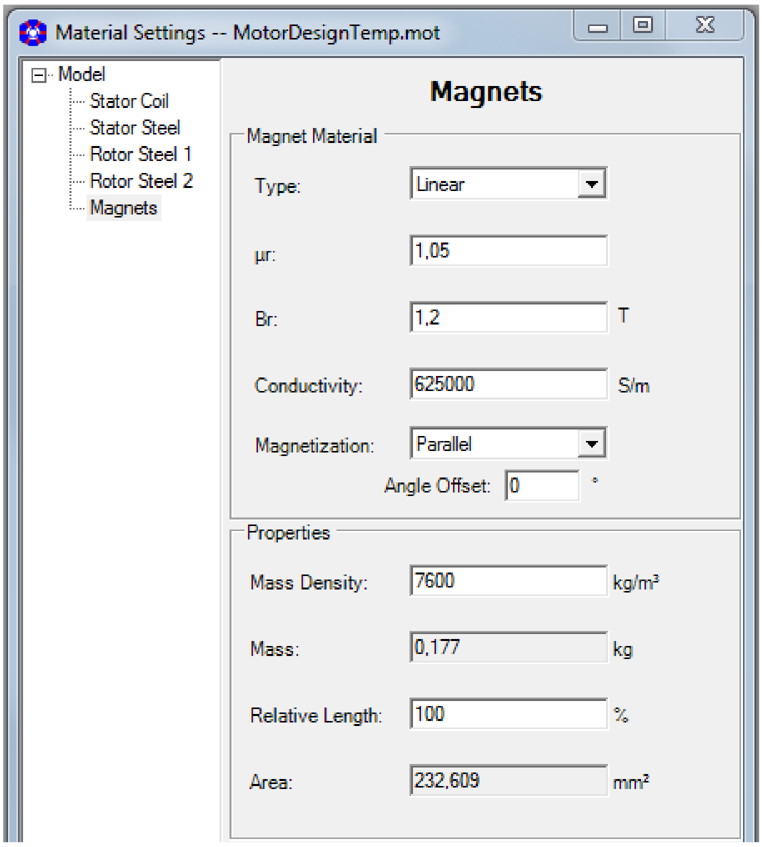


Fig. 41: Material Settings - Magnets

In case of *“Non-Linear”* calculation can the demagnetization characteristic curves and loss coefficients also be defined with the *“Material Explorer”* and selected in *„Material“*.

The magnetization direction can be set on "*Parallel*", "*Radial*", "*Halbach*" and "*User Defined*" whereby the direction of magnetization is displayed in the graphical window.

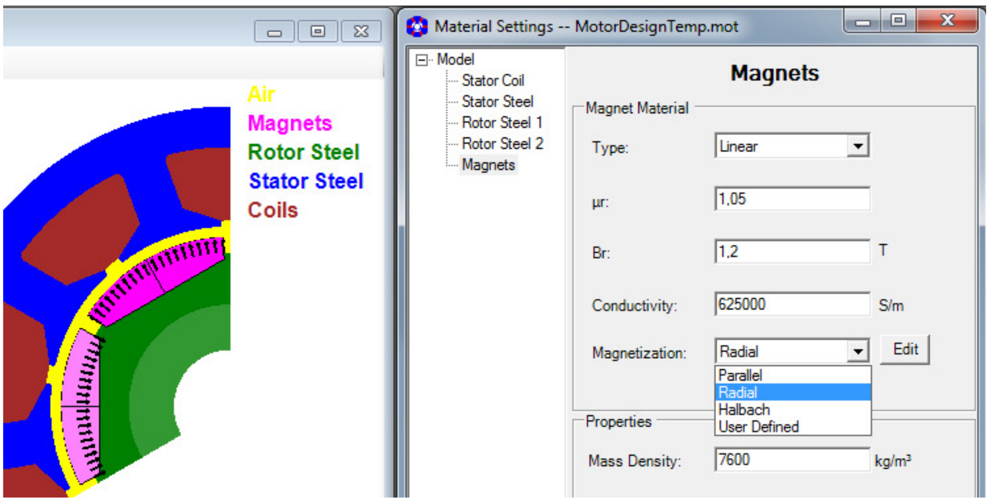


Fig. 42: Material Settings - Magnetisation

For *"Parallel"* magnetization an offset angle for rotation of the magnetization direction can additionally be specified.

In case of *"Radial"* the magnetization can be additionally edited in a text table, displayed in a graphical window and saved as text file *.dat in the material folder. The magnetization is automatically adjusted to the respectively selected magnetic width (W_m = 180 °el).

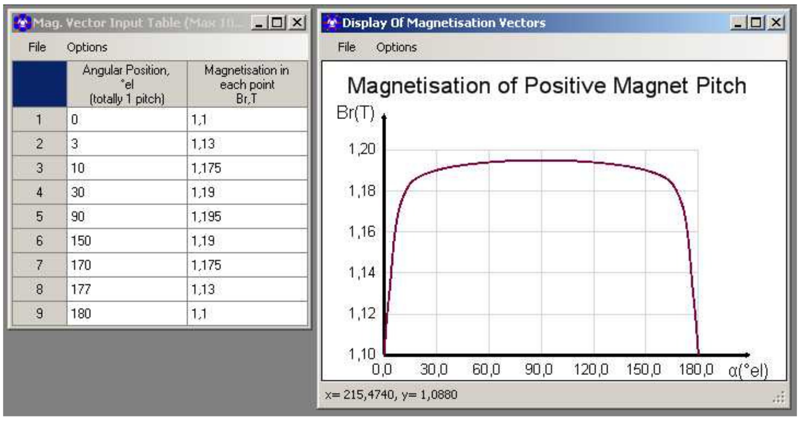


Fig. 43: Material Settings - Magnetization table

In "*User Defined*" magnetization, for each individual magnet, all parameter shown in the text table can be edited and set as "*Default*".

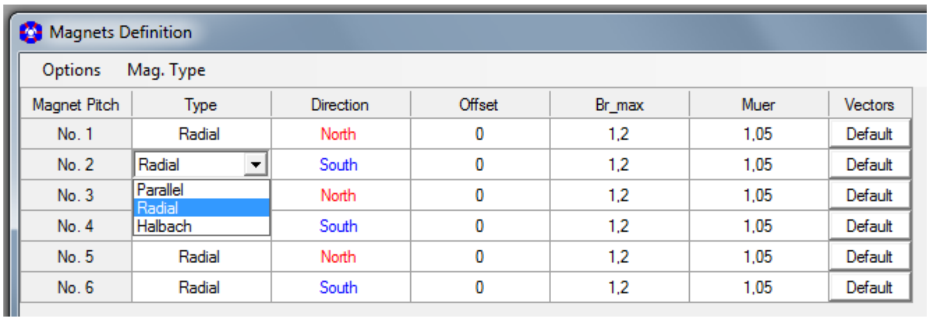


Fig. 44: Material Settings - User defined magnetisation

There is also a possibility of importing measured magnetization of ring magnets and converting it for the use in smartFEM and FEMAG.

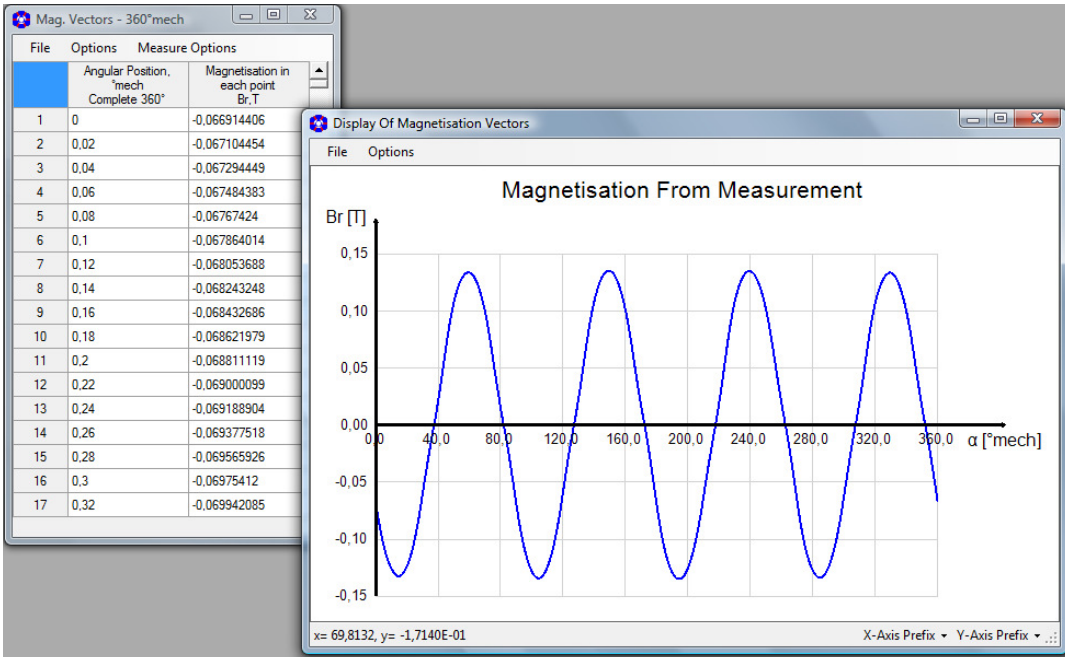


Fig. 45: Material Settings - Measured magnetization

Additional parameters have to be entered in the menu "*Measure Options*" to complete the measurement data.

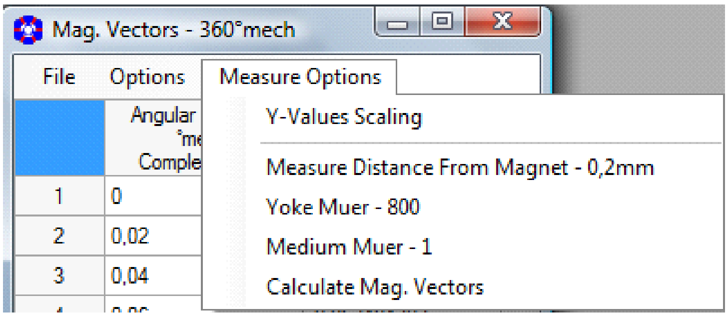
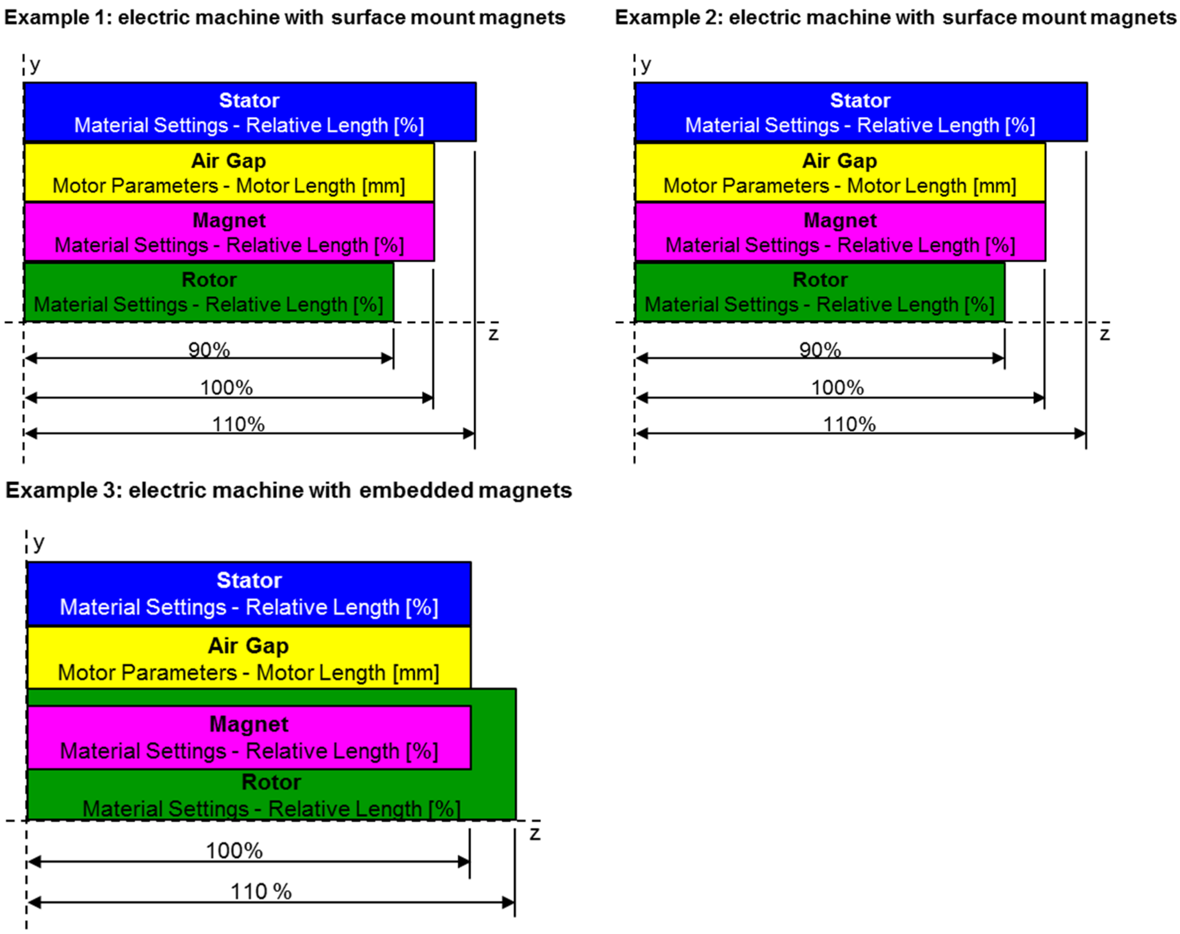


Fig. 46: Material Settings - Additional parameters for measured magnetization

3.2.4 Relative Length

For permanent magnets and soft iron can different magnetic effective length of the materials in axial direction (z-axis) be taken into account whereby a constant flux is assumed. Reference value is the **axial length of the air gap = 100%**.



For windings is the value of the *"Relative Length"* taken into account for calculation of the ohmic resistance. Furthermore is the *"Relative Length"* taken into account for the calculation of the mass of all materials.

3.2.5 Material Explorer / Material Editor

The material properties of electrical steel and permanent magnets can be edited and viewed with the smartFEM *"Material Explorer"*. It includes an interface to convert the material data into formatted files which can be used for FEM calculations with FEMAG, JMAG, MAXWELL, SPEED and into formatted text files for individual use in other software tools.

The Material Explorer was developed in collaboration with thyssenkrupp Steel Europe AG (TKSE) in Bochum, Germany. TKSE provides the software named as *"PowerCore® Explorer"* independently from smartFEM.

The functionality of Material Explorer is described in the document "*User Guide Material Explorer*".

3.2.5.1 Electrical Steel

With installation of smartFEM, the data of three TKSE standard materials for non-linear calculations are copied into the smartFEM material folder. Further material data of TKSE product range can be received directly from TKSE in Bochum.

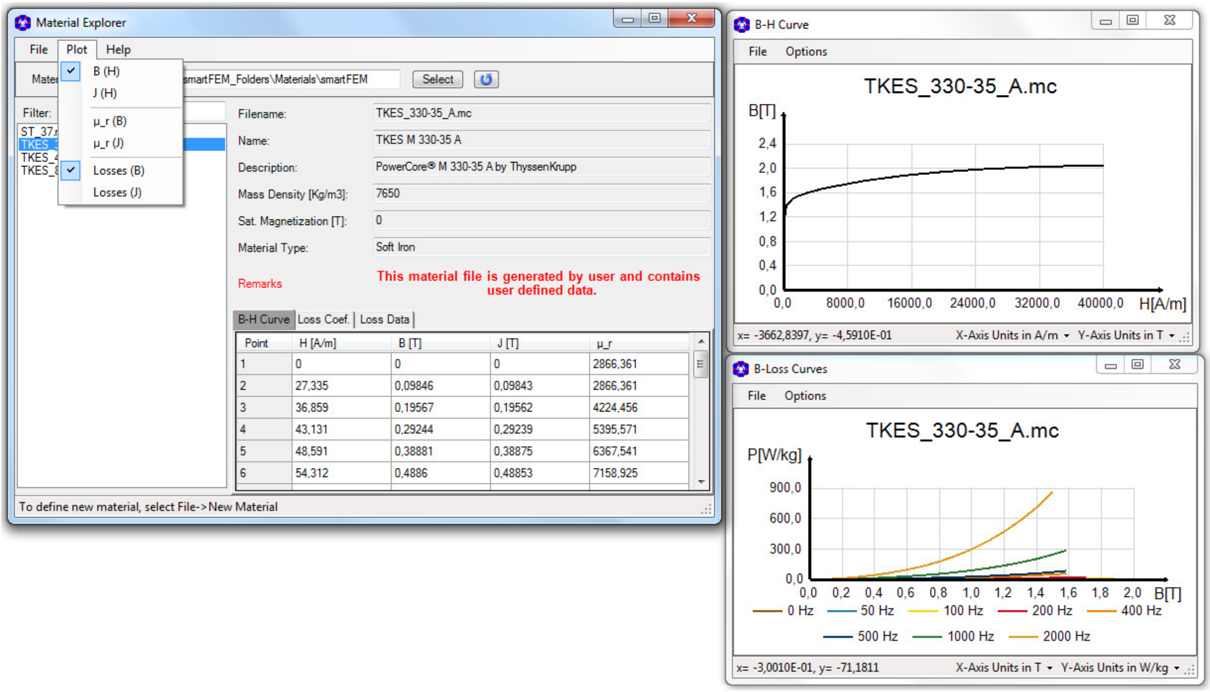


Fig. 47: Material Explorer - B(H) and loss data of electrical steel

3.2.5.2 Permanent Magnets

In addition to electrical steel, with the help of Material Explorer also non-linear "demagnetization characteristics curves" of permanent magnets can be recorded and can be stored as FEMAG specific mc-, mca- and txt-files. For this purpose, the material type "*Permanent Magnet*" must be selected when storing the files.

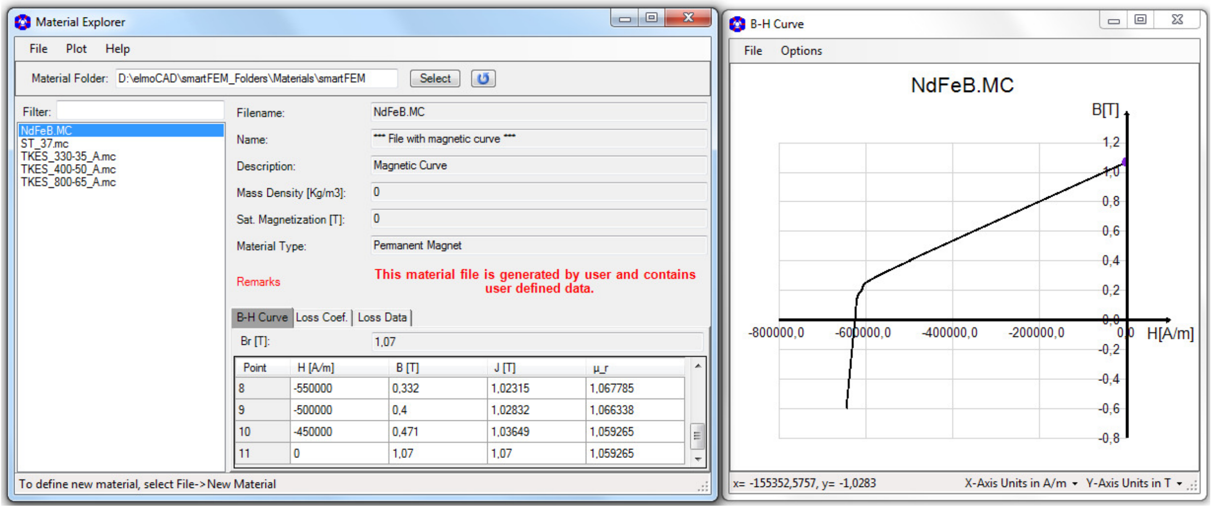


Fig. 48: Material Explorer - Demagnetization characteristics of magnet material

At present, the demagnetization characteristics curves of a material for different temperatures have to be individually recorded and stored.

Loss coefficients and loss data are currently not used by smartFEM. To calculate the eddy current losses in the magnets, the conductivity entered in *"Materials"* is used.

The creations of the characteristics for <u>non-magnetized</u> magnetic material are analogous to the characteristics for electrical steel. This data is used in the simulation of magnetization processes with the smartFEM module *„PM - Magnetization - Inner Rotor".*

3.3 Preprocessing

A prerequisite for the calculation of results and their analysis by FEM simulation is the structure of the FEM model in FEMAG. After pressing the *"Preprocessing"* button, FEMAG is started, all necessary information about geometry, node chains and materials is transferred to FEMAG, and the FEMAG process from smartFEM to the field calculation is controlled by the FEMAG menu sequence *"Field Calculation" → "Calculate Once".*

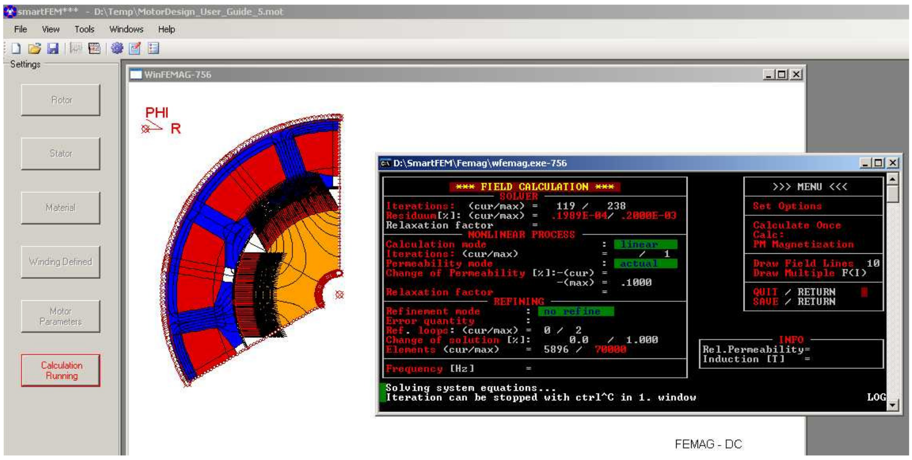


Fig. 49: Preprocessing - Structure of the FEM model with FEMAG

After successful *"Preprocessing",* the button *"Calculation"* will be displayed. The user can now carry out a first FEM simulation with determination of the cogging torque (if applicable at this time) (see 3.6.1 Cogging Torque and BEMF).

The *"Preprocessing"* can be controlled with right-click on the *"Preprocessing"* button as follows:

- Show / Hide FEMAG	Show / Hide FEMAG windows.
- Stop FEMAG on Error	FEMAG is paused after an error, so that the user can determine the cause of the error.
- Only Meshing	Only meshing is performed.
- Preprocessing + Calculation	After *"Preprocessing",* the calculation type selected in Motor Parameters is automatically executed.

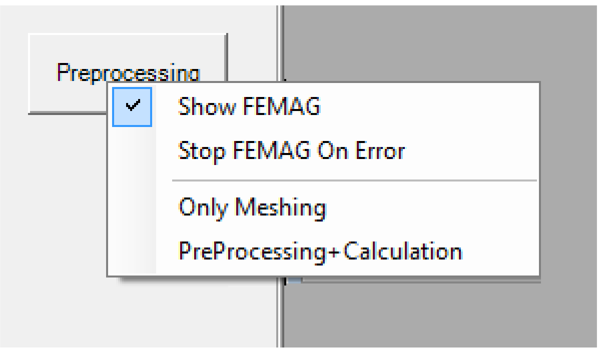


Fig. 50: Preprocessing - Show FEMAG

After *"Meshing"* has been successfully completed, the network representation can be opened via the *"Plots"* button.

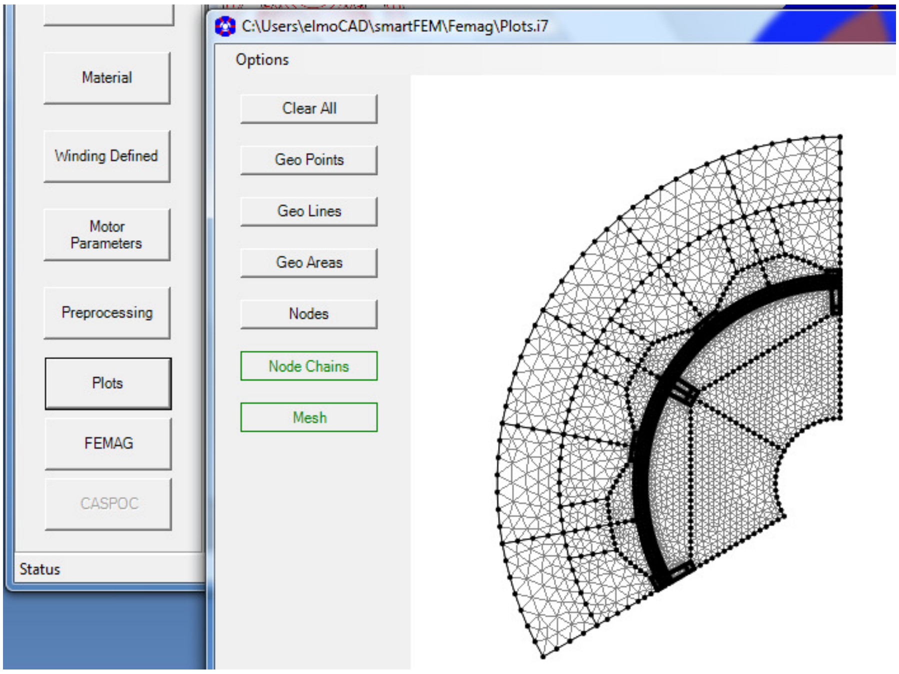


Fig. 51: Preprocessing - Meshing and node chains

"*Preprocessing*" can be completed after optimizing the mesh by further pressing the "Preprocessing" button.

As soon as a change to motor geometry or material is made any later time then the text *"Calculation"* is automatically reset to *"Preprocessing"*, all results are deleted and the preprocessing must be run again.

For the calculation of further results e.g. BEMF, Torque, etc., the motor model has to be completed together with information on windings and further motor parameters (see 3.4 *Plots* and 3.6 *Motor Parameters*).

The calculation of the field distribution in the inner region of the rotor (region < inner rotor radius Rri) can be performed on complete machine models (360°mech). It can be selected in ***"Motor Parameters – Options – Calculate and Mesh Shaft"***.

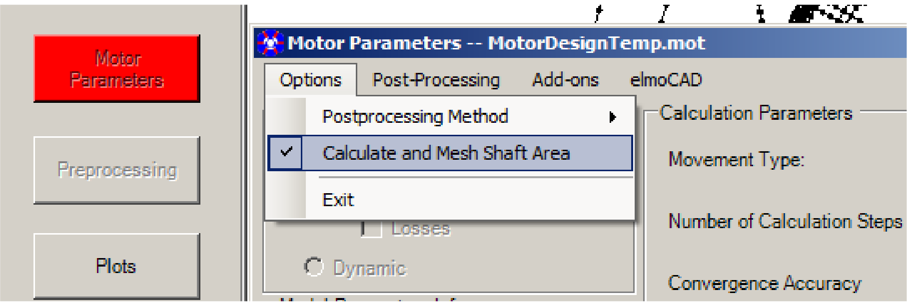


Fig. 52: Preprocessing - Calculate and Mesh Shaft Area

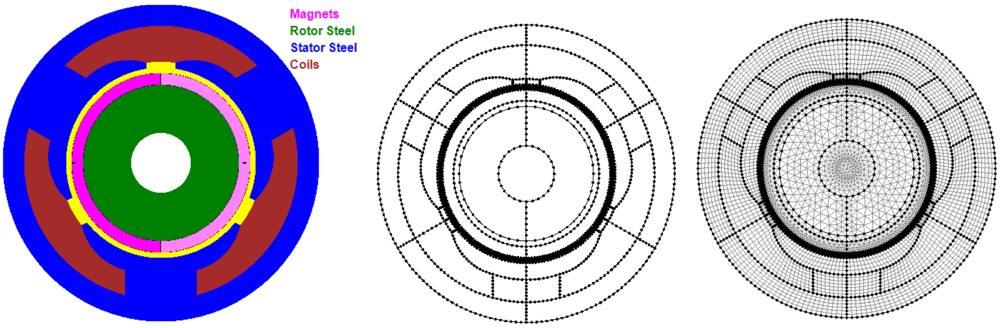


Fig. 53: Preprocessing - Calculate and Mesh Shaft Area (2)

This type of meshing is only possible for complete machine geometries (CompleteGeometry=true) because in all other cases boundary conditions are required by Femag.

Remark: the field distribution is be calculated without this inner mesh wrong for 2-pole machines!

incorrect field distribution

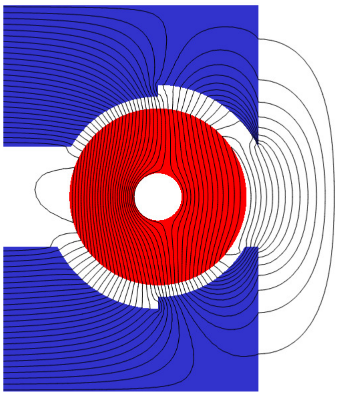


Fig. 54: Preprocessing - incorrect field distribution

proper field distribution

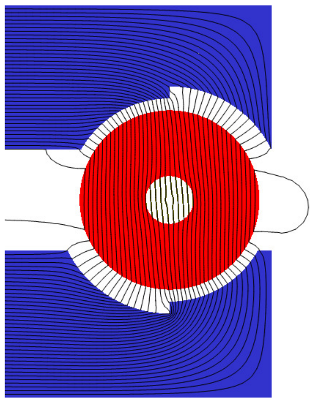


Fig. 55: Preprocessing - proper field distribution

3.4 Field Plots

3.4.1 Definition and display of field plots

After successful *"Preprocessing"*, the design of the magnetic circuit can be assessed using *"Plots"*. To do this, in the *"Field Plots Definition"* menu, *"Plots - Define Picture"* must be opened:

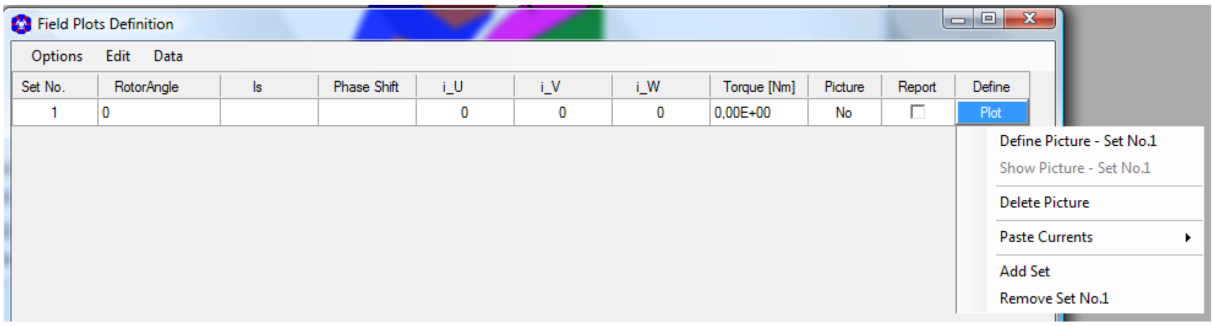


Fig. 56: Field Plots - Field Plots Definition

After opening the plot window (it is possible that FEMAG field calculations are performed beforehand), the various plots can be selected and displayed:

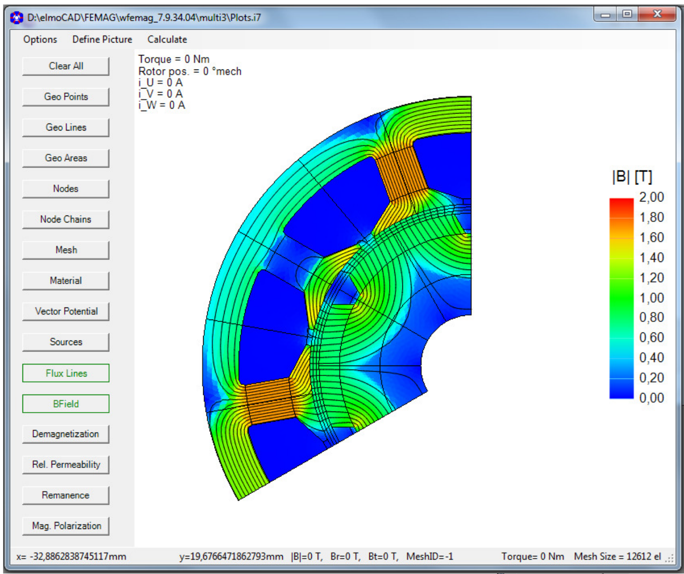


Fig. 57: Field Plots - Magnetic induction with superimposed flux lines

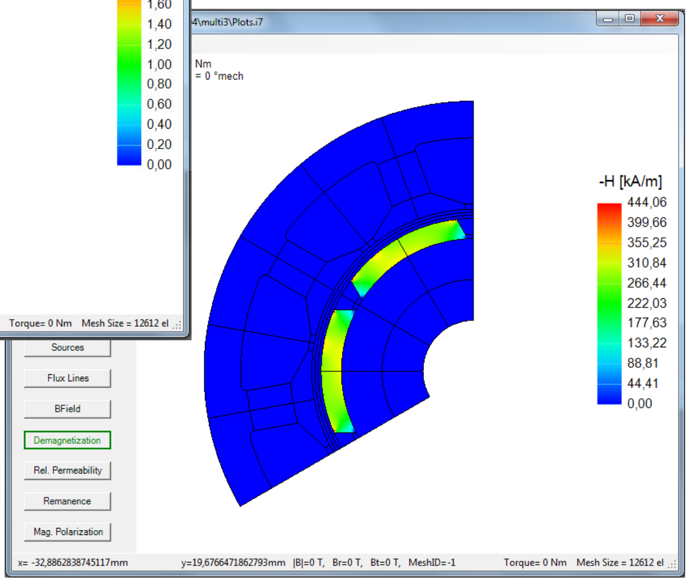


Fig. 58: Field Plots - Demagnetization

Flux lines, scaling and other parameters can be adjusted by clicking the right mouse button on the selection field.

The plots can be viewed on the screen as well as on the project report. All plots can be overlaid. The overlay is undone by a further click on the corresponding button. The number of flux lines as well as the minimum or maximum values of the result calculations can be adjusted by clicking the corresponding button with the right mouse button.

Furthermore, details can be viewed at any depth using the already mentioned zoom function. In addition, the x/y coordinates of the mouse pointer and other information are displayed in the status line.

The user can select from the following plots:

• Geometry
  - Points
  - Lines
  - Areas

• FEM-base
  - Node
  - Node chains
  - Networking
  - Materials

• Magnetic circuit results
  - Vector potential
  - Current densities
  - Flux lines
  - Magnetic flux density
  - Demagnetization
  - Relative permeability
  - Remanence
  - Magnetic polarization

A bigger number of sets can be created easily by using the “*Parametic”* function.

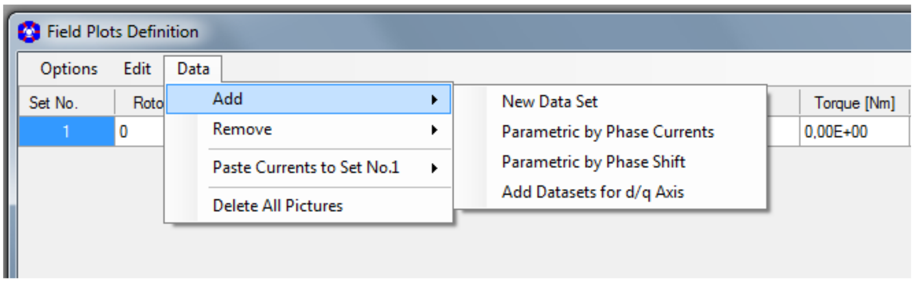


Fig. 59: Field Plots - Table for parametric definition of sets

3.4.2 Model Data

For user-defined evaluation, the properties of nodes and network elements can be exported via the clipboard:

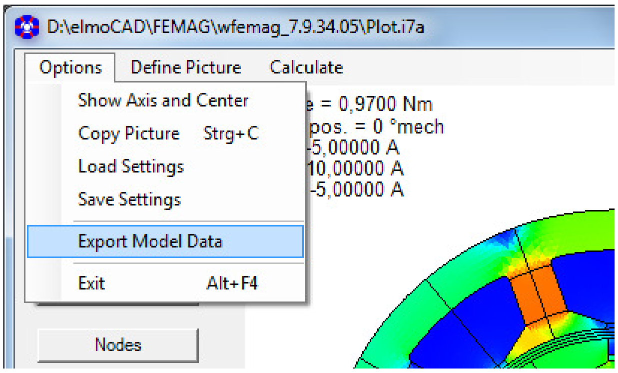


Fig. 60: Field Plots - Export Model Data

NODES PROPERTIES								
Nr.	X [mm]	Y [mm]	Rad [mm]	Angle [rad]	Vre [V/m]			
1	-2,55E-11	4,999999888	4,999999888	1,570796371	0			
2	-0,52264228	4,972609226	4,999999888	1,675516129	0			
3	-1,039557974	4,890738055	4,999999888	1,780235767	0			
4	0	8,999999613	8,999999613	1,570796371	-0,000953574			
5	-0,2617625	8,996193297	9,000000544	1,599885106	-0,000947984			
...	...	...	...	...	...			
...	...	...	...	...	...			
MESH PROPERTIES								
Nr.	Material	Area[mm2]	CenterR[mm]	1terAngle[°mech]	Br[T]	Bt[T]	Bs[T]	Nodes
1	RotorYoke	0,094855711	5,114084	146,6292893	0,001113491	-0,172096813	0,172100415	(783 48 47)
2	RotorYoke	0,094855713	5,282548	148,6280981	-0,000379586	-0,17678462	0,176785028	(783 526 48)
3	RotorYoke	0,053620981	8,740101	149,5313127	0,028053478	-0,751927743	0,752450881	(784 43 532)
4	RotorYoke	0,036313581	8,906621	148,9787244	0,023961995	-0,760617302	0,76099465	(784 42 43)
5	RotorYoke	0,036315428	8,906616	91,02122319	0,023654246	0,700147637	0,700547099	(785 7 8)
...	...	...	...	...	...			
...	...	...	...	...	...			
1383	Magnet1	0,027852678	13,00234	146,8191891	1,020952942	-0,461262713	1,120316116	(1386 574 575)
1384	Magnet1	0,027852653	13,12514	147,1621962	1,021633191	-0,473463444	1,126011549	(1386 620 574)
1385	Magnet1	0,021788364	14,85829	147,2623732	0,636300937	0,265874223	0,689614374	(1387 103 619)
1386	Magnet1	0,023175448	14,94144	146,9229567	0,426899132	0,059100694	0,43097072	(1387 102 103)
1387	Magnet1	0,023175551	14,94144	93,07703825	0,36248814	-0,116518681	0,380754848	(1388 49 50)
...	...	...	...	...	...			
...	...	...	...	...	...			
2719	Air	0,054310899	14,86109	90,41919031	-0,028352828	-0,236666066	0,238358363	(640 641 642 1995)
2720	Air	0,053398312	14,61132	90,41919034	-0,034010782	-0,163672559	0,167168896	(639 640 1995 1996)
2721	Air	0,052485521	14,36155	90,41919036	-0,029746282	-0,08789884	0,092795729	(638 639 1996 1997)
2722	Air	0,05157273	14,11179	90,4191904	-0,019442563	-0,028442925	0,034453059	(637 638 1997 1998)
2723	Air	0,050659939	13,86202	90,41919043	-0,00668296	0,003608685	0,007595036	(636 637 1998 1999)
...	...	...	...	...	...			
...	...	...	...	...	...			
2755	StatorYoke	0,089996783	22,44139	109,2922672	1,770401525	-0,049111997	1,771082592	(2015 659 662)
2756	StatorYoke	0,089996749	22,61428	109,7093078	1,698753324	0,003731617	1,698757423	(2015 658 659)

Fig. 61: Field Plots - Exported model data example

Through "Calculate", the torque, radial force and the angle of the force vector in the air gap can be calculated. At the same time, the data is made available in the buffer memory.

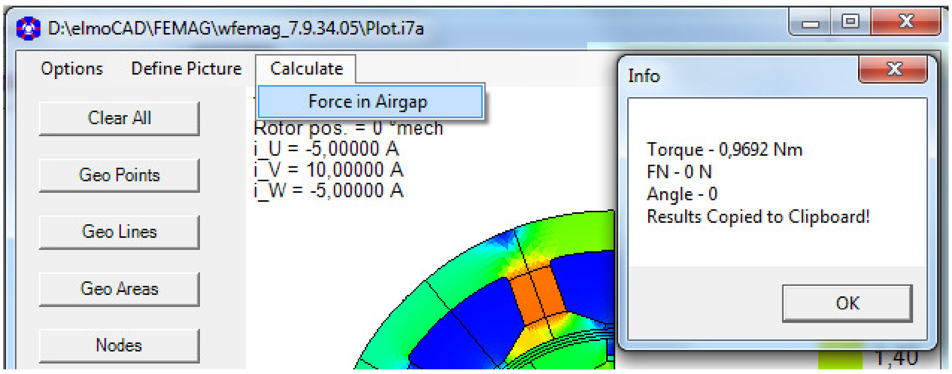
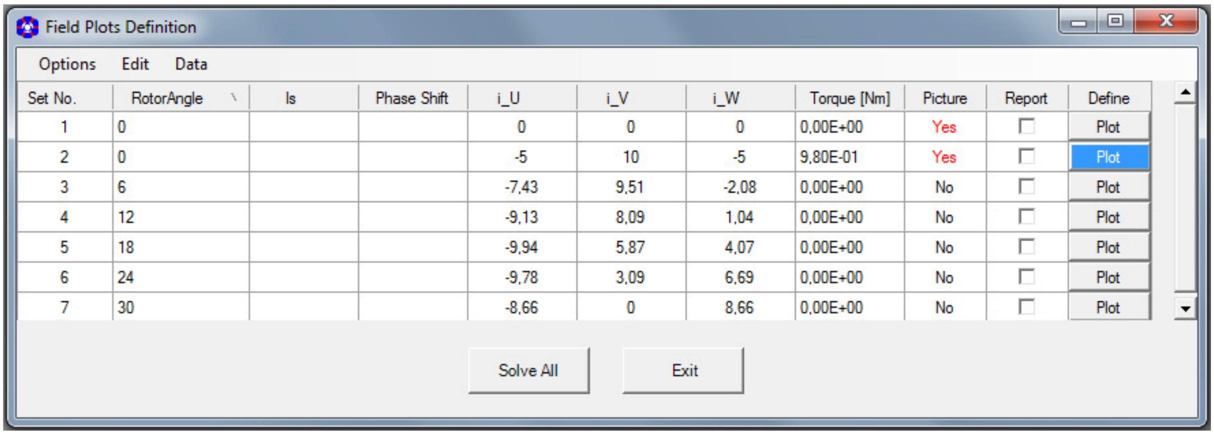


Fig. 62: Field Plots - Force in Air gap

3.4.3 Creation of Filed Plots with the same layout

In the *"Field Plots Definition"* table several images can be defined for different rotor positions and currents. They can be defined individually with *"Define"* → *"Picture"* using different contents. Currents can be entered manually or transferred from the results of *"Expected values"*, *"Nominal Torque"* or *"Dynamic"* for the selected table lines through the *"Data"* → *"Paste Currents"* dialogue ".



Set No.	RotorAngle	Is	Phase Shift	i_U	i_V	i_W	Torque [Nm]	Picture	Report	Define
1	0			0	0	0	0,00E+00	Yes		Plot
2	0			-5	10	-5	9,80E-01	Yes		Plot
3	6			-7,43	9,51	-2,08	0,00E+00	No		Plot
4	12			-9,13	8,09	1,04	0,00E+00	No		Plot
5	18			-9,94	5,87	4,07	0,00E+00	No		Plot
6	24			-9,78	3,09	6,69	0,00E+00	No		Plot
7	30			-8,66	0	8,66	0,00E+00	No		Plot

Fig. 63: Field Plots - Definition of sets

The images can also be created with "Solve All". In this case all the images show the same section. However, to achieve this, the settings of an image must be saved beforehand via "Options" → "Save Settings" as well as the image itself with "Define Picture".

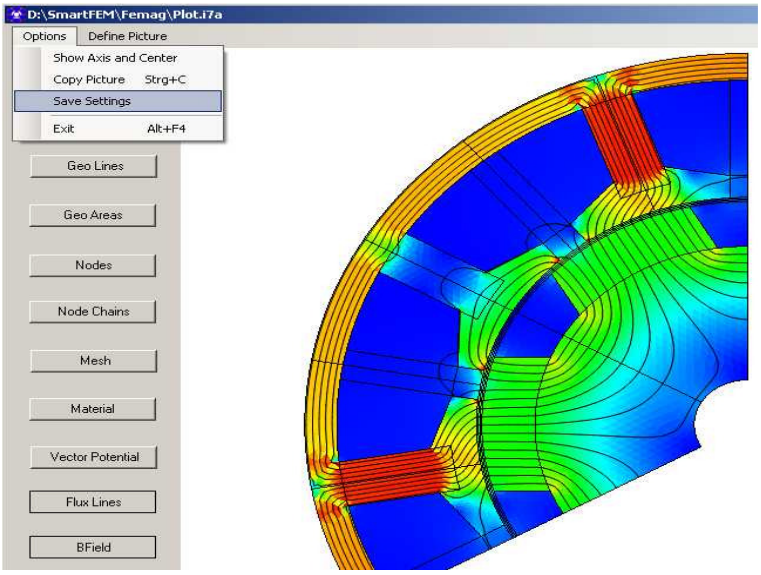


Fig. 64: Field Plots - Saving settings after crating the image content

If then an image with *"Show Picture"* is opened and the arrow keys are used to move up or down in the *"Field Plots Definition"* table, all images are displayed successively, so that the field changes can be observed very well. By clicking on a column header, the table is sorted up or down according to the column values.

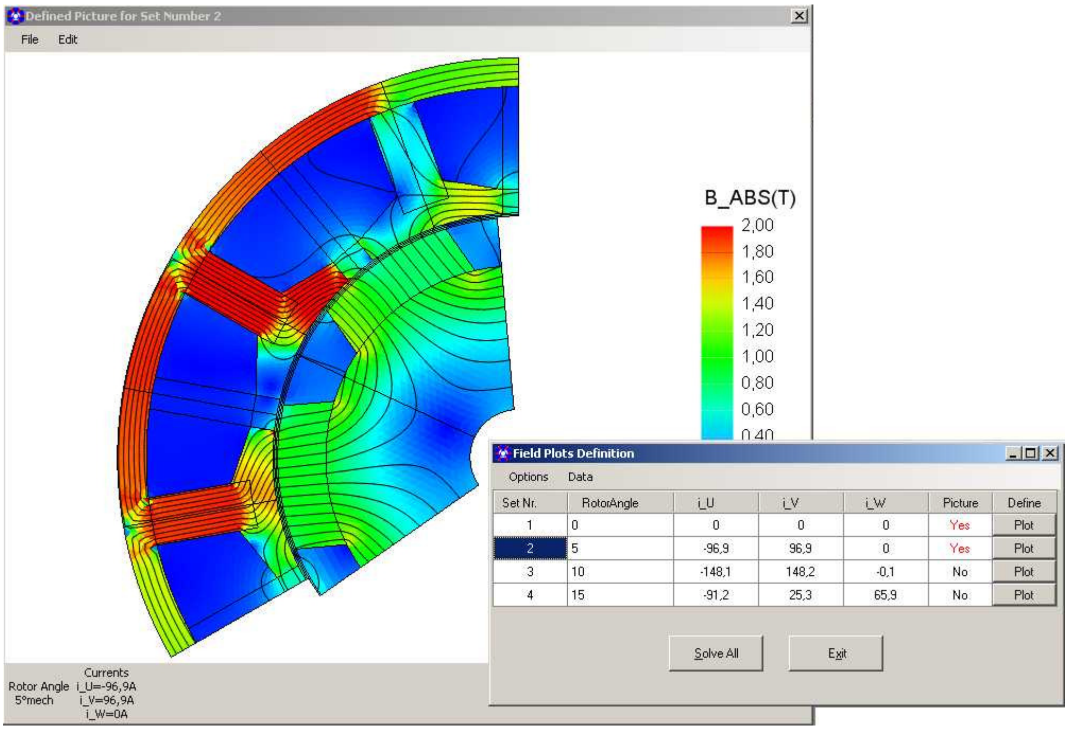


Fig. 65: Field Plots - Display of the plots per set

Flux lines can be displayed in different colors (black/white is default). By clicking the right mouse button on the "Flux Lines" button, a corresponding pop-up window opens.

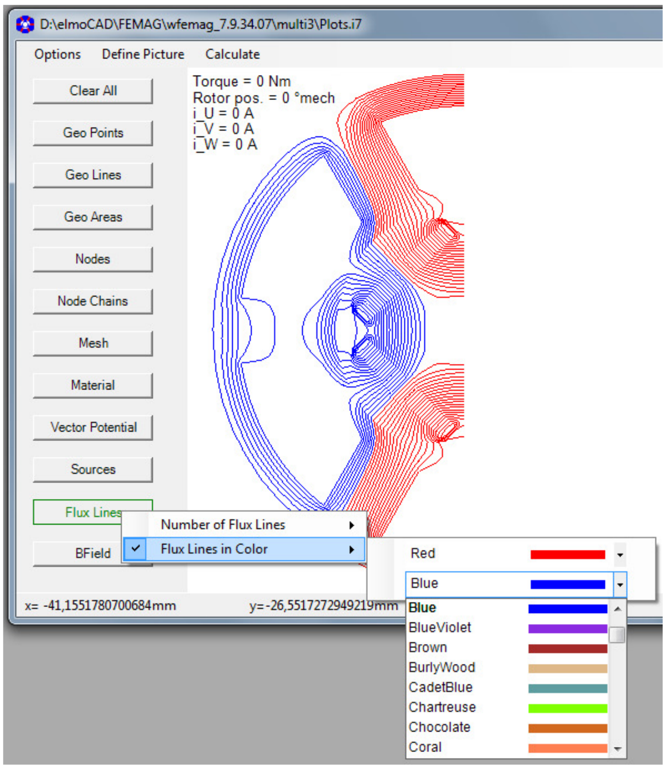


Fig. 66: Field Plots - Colored flux lines

In the smartFEM settings, the colors for the direction of the flux lines can be adjusted clockwise and counterclockwise.

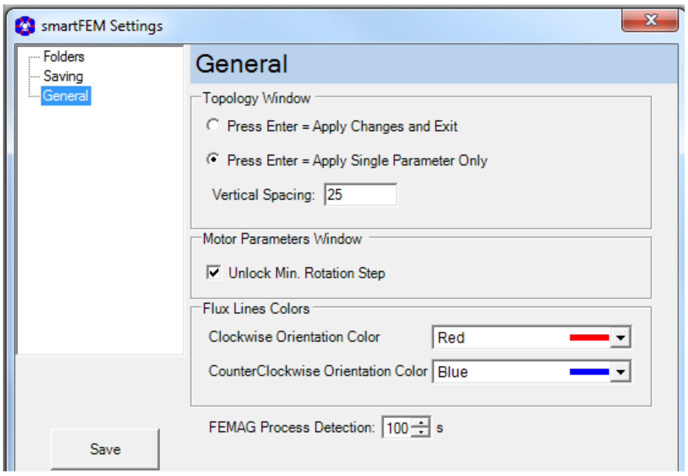


Fig. 67: Field Plots - Colored flux lines

The table *„Field Plots Definition“* can be copied into the clipboard containing all calculation results.

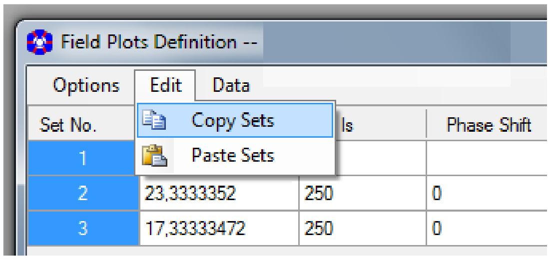


Fig. 68: Field Plots - Copy Results

	A	B	C	D	E	F	G	H	I	J	K
1	RotorAngle [	Is [A]	Theta [°el]	i_U [A]	i_V [A]	i_W [A]	Torque [Nm]	Flux_U [Vs]	Flux_V [Vs]	Flux_W [Vs]	T/Is [Nm/A]
2	0	250	0	-43,41	-191,5	234,9	1274,02588	0,203206	-0,247091	0,0518997	5,0961
3	23,3333352	250	0	0,0002947	-216,5	216,5	1270,30273	0,232939	-0,226686	0,00748404	5,0812
4	17,3333347	250	0	250	-125	-125	1296,38147	0,1333	0,1258	-0,2672	5,1855

Fig. 69: Field Plots - Table containing calculation results

3.5 Windings

For the description of windings, there is a winding editor available at smartFEM. If there is at least one permissible winding scheme for the motor geometry described up to this point, the button *"Winding Defined"* is displayed and the winding scheme is set as default. Otherwise the text *"Winding Not Defined"* is displayed in the button. N-phase winding patterns can be described as single/two-layered with integral/fractional windings and parallel winding groups/phase.

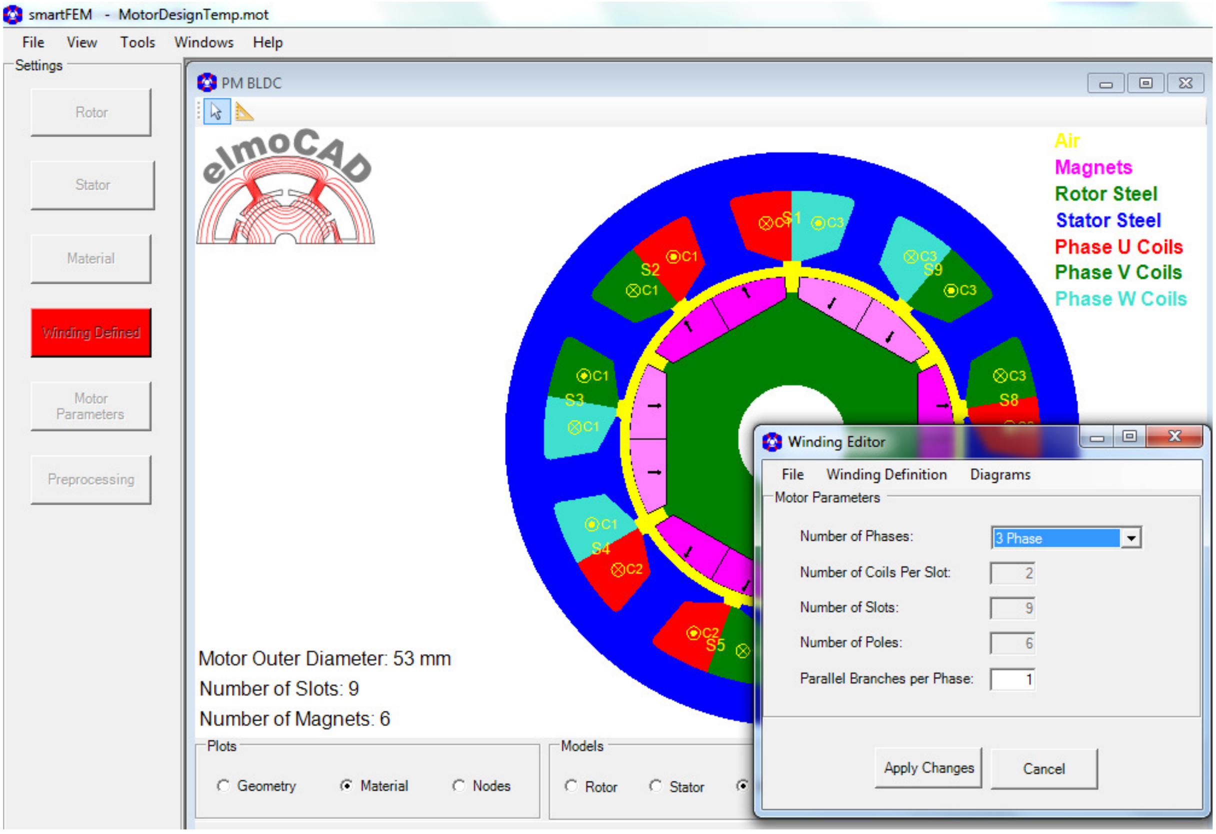


Fig. 70: Winding Editor - Opening the smartFEM button "Winding ..."

The number of slots (S ..) and individual coils (C ..) together with current direction (in/out) are displayed in the winding surfaces.

The color and the size of the text and icons can be set in "*smartFEM Settings - GUI*".

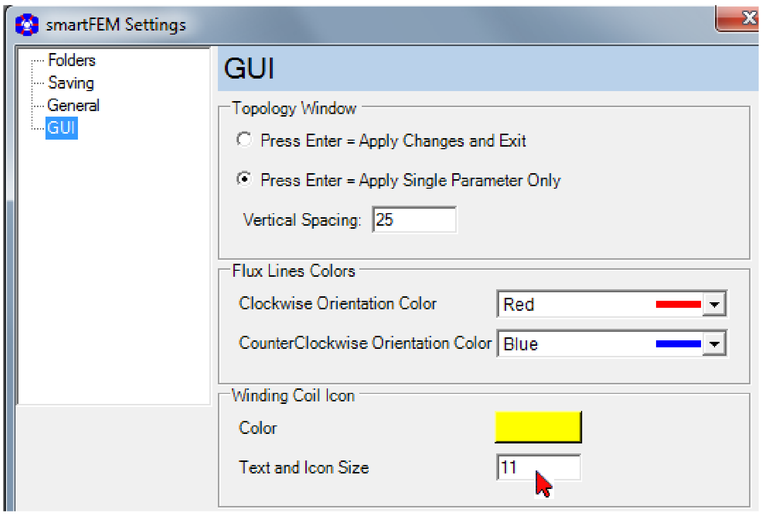


Fig. 71: Winding Editor - The color and the size of the text/icons in smartFEM Settings

3.5.1 Coils Parameters

Through *"Winding Definition"* → *"Coils Parameters"* a pop-up window will be opened in Which input fields the wire diameter, number of turns per coil and an additive phase resistance/induction can be entered.

For a conductor type, "round" or "rectangular" can be selected.

In addition, the slot area available for the winding per slot and the fill factor are displayed. The fill factor is calculated as follows:

Fill Factor = Number Of Turns per coil *
Conductor Area / Slot Area

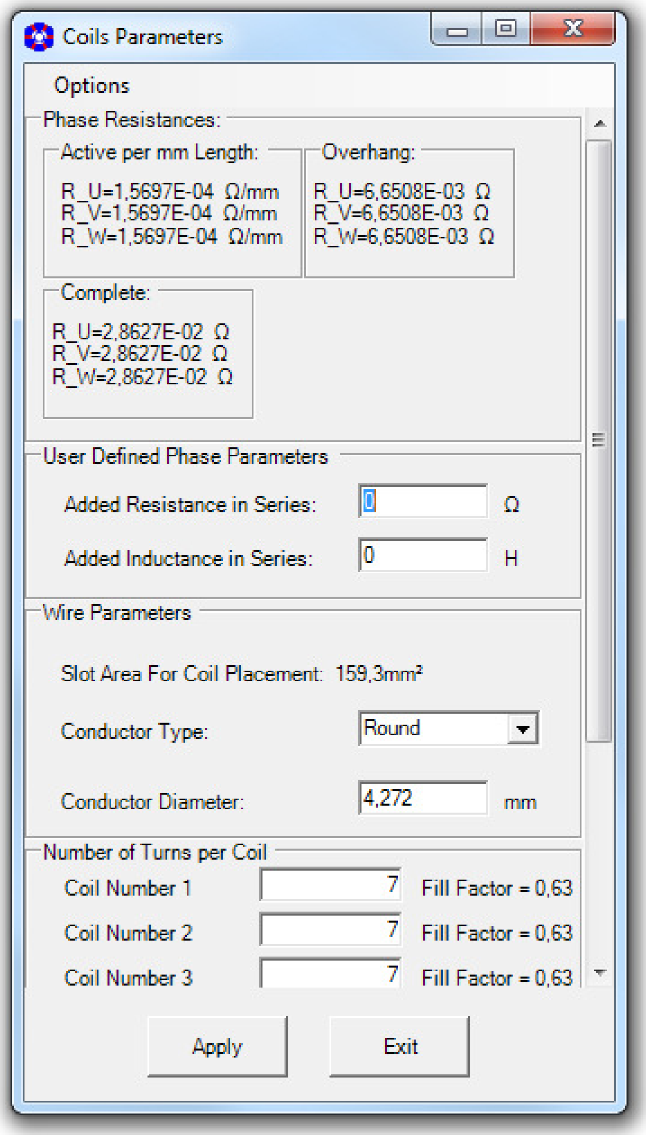


Fig. 72: Winding Editor - Coils parameters

With *"Options"* another menu is displayed. It provides functions for specifying the number of turns, calculating the number of turns at a predetermined filling factor and calculating the conductor cross section with predetermined values for fill factor and number of turns available.

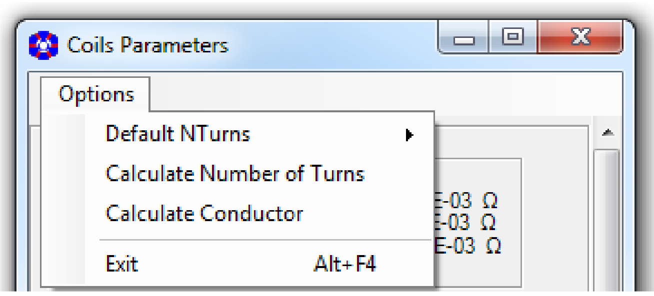


Fig. 73: Winding Editor - Coils parameters - Options

3.5.2 Winding Scheme

Through *"Winding Definition"* → *"Winding Scheme",* the winding scheme can be processed in a list. As additional support, the entire motor is displayed as a graph. As soon as there is a permissible winding scheme, it is displayed as *"Auto Winding".* The slots corresponding to the phases are colored. The slot area in which the cursor is located is shown in yellow. With the help of *"Auto Winding"* the winding scheme is generated, with the maximum induced voltage.

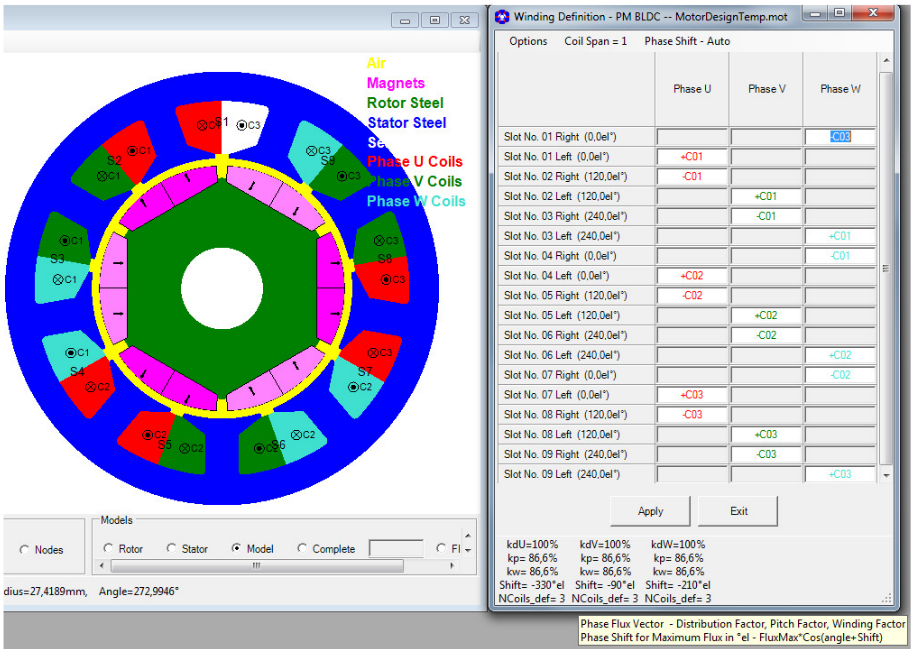


Fig. 74: Winding Editor - Winding scheme "two-layer fractional slot winding" The values for amplitude Flux Vector, Distribution, Pitch and Winding factor are displayed

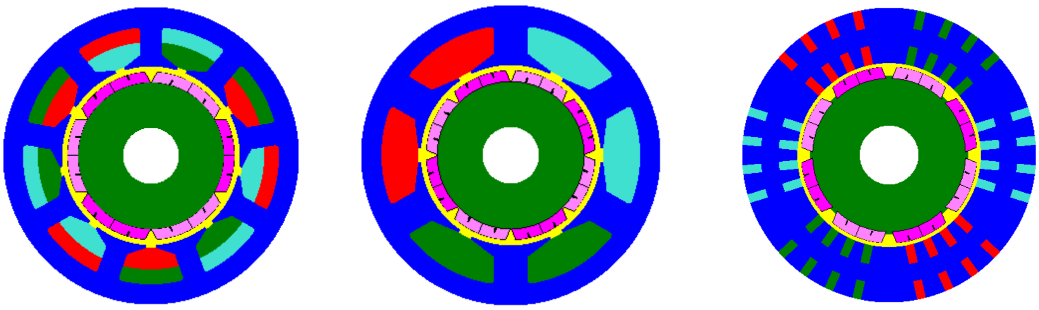


Fig. 75: "Two-layered fractional slot, "One-layered winding" "Ring winding"

„Options":

- *"Default Winding":* The winding scheme with which the window was initially opened, appears again.
- *"Auto Winding":* The winding scheme with maximum BEMF will be automatically determined and displayed.
- "*Symmetric Winding":* The input for the phase U is automatically assigned into the V and W phases with an offset of + 120 ° el.
- *"Clear All":* The winding pattern is deleted and can then be redefined by the user. By clicking in an entry field from the list, are not yet specified windings shown and can be selected. The corresponding slot surface is then filled with the color of the respective phase.

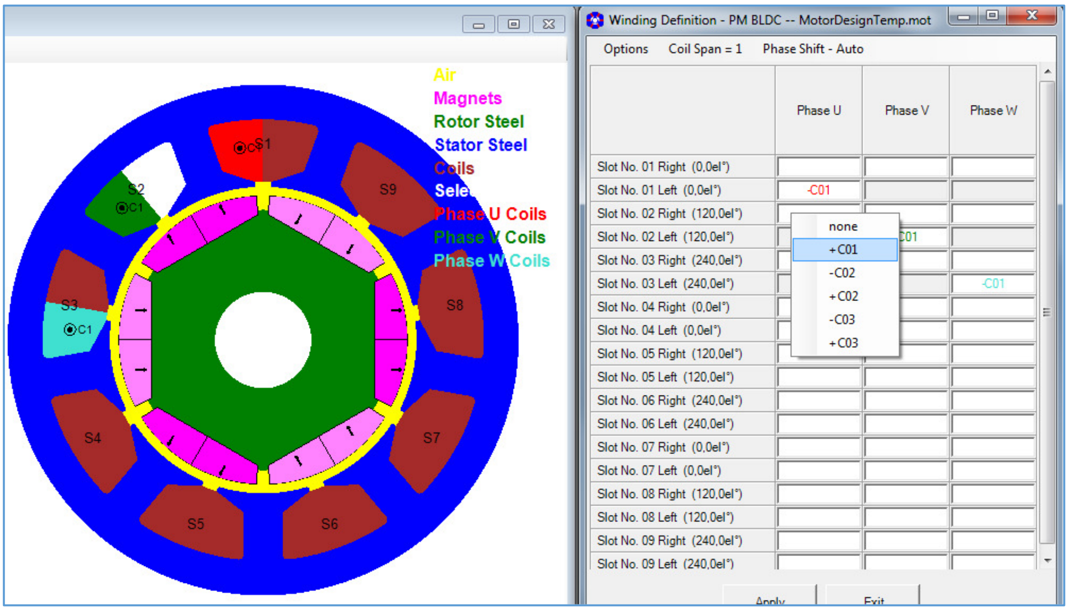


Fig. 76: Winding Editor - User-definition of the winding scheme

A user-defined winding scheme is adopted by clicking the *"Apply"* button. The entire winding definition can be saved with *"File"* → *"Save Winding Definition"* in a text file *.wdf and later used in other motor models through *"File"* → *"Open Winding Definition"*.

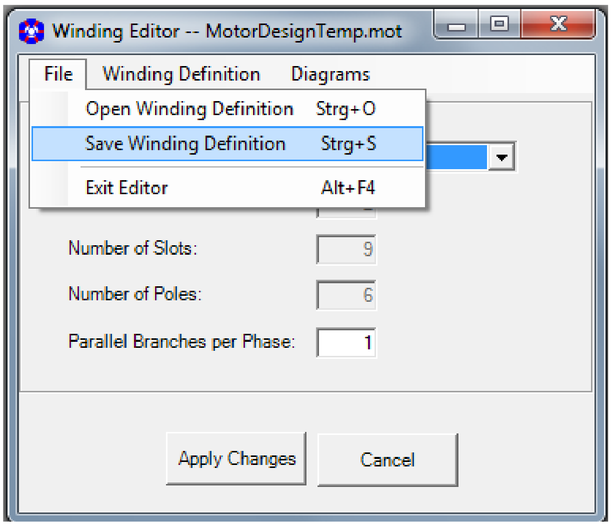


Fig. 77: Winding Editor - Saving or loading a winding definition

The winding editor is closed with *"Apply Changes"*, in case that the winding definition should be adopted for the current motor model. If the current motor design should not be changed, the winding editor has to be closed via *"Cancel"*.

3.5.3 Phase Diagram

Through the *"Diagrams"* menu, the vectors of the phases and coils are displayed. With this is the user able to check the winding scheme for faults.

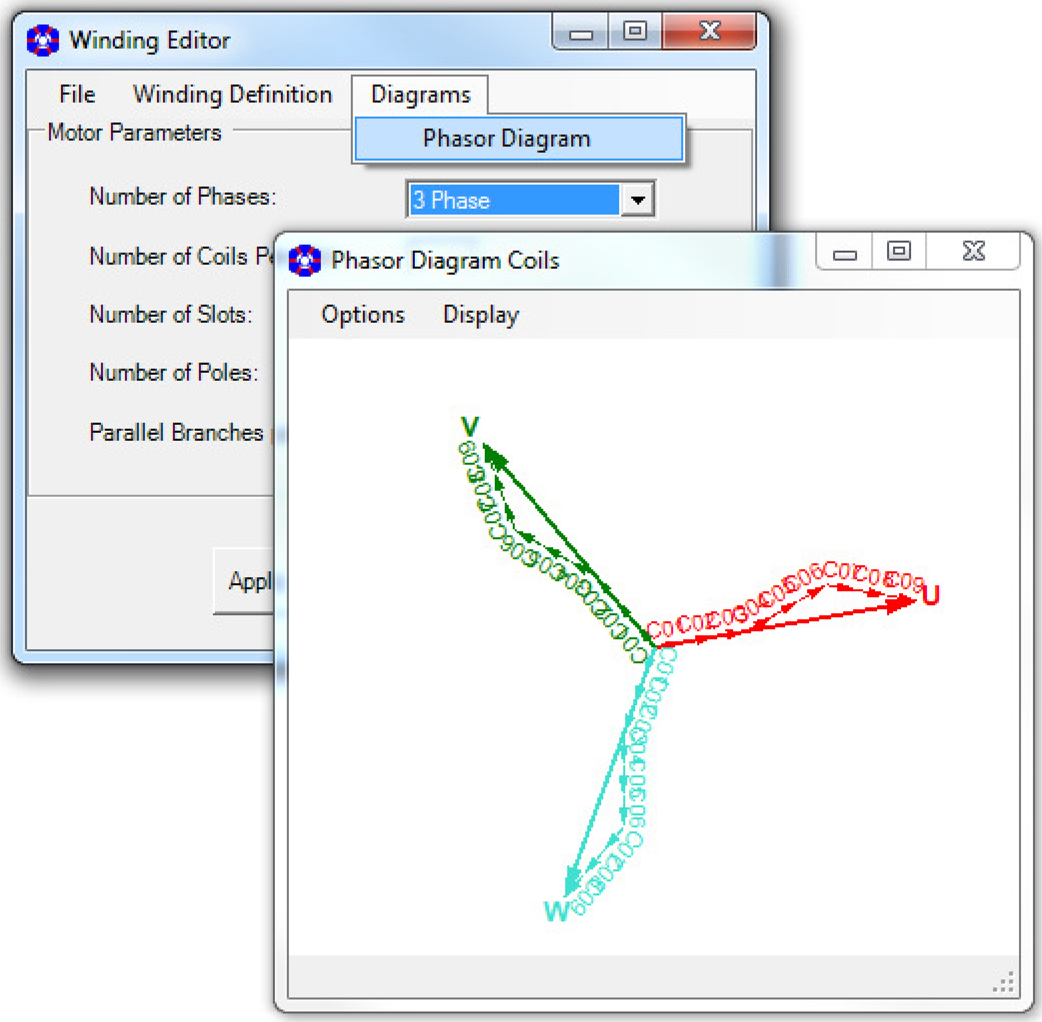


Fig. 78: Winding Editor - Phasor Diagram

3.6 Motor parameters and calculations

Depending on simulations and calculations the user wants to carry out, various motor parameters must be entered.

All simulations of the motor are related to rotor positions. Starting from the normal 0° position of the rotor is it rotated counterclockwise by a certain angle after each calculation in order to perform the next simulation. The number of calculation steps and the calculation accuracy can be specified as a termination condition by the user.

The calculations are triggered via the *"Calculation"* button. By clicking with the right mouse button on the *"Calculation"* button, various settings can be made in accordance to the "*Preprocessing"* button.

3.6.1 Base for the results calculations

The base for the results calculation is the single-phase equivalent circuit diagram of a synchronous motor.

The in *“Results”* displayed *"Torque With Applied Current"* represents the "inner" torque because iron, magnet and other losses which lead to a reduction of the torque on the shaft is not taken into account during FEM calculations. These losses can be calculated as a torque loss for the given speed with the help of the formula P = T * ω and will be considered in *"Torque After Losses“*.

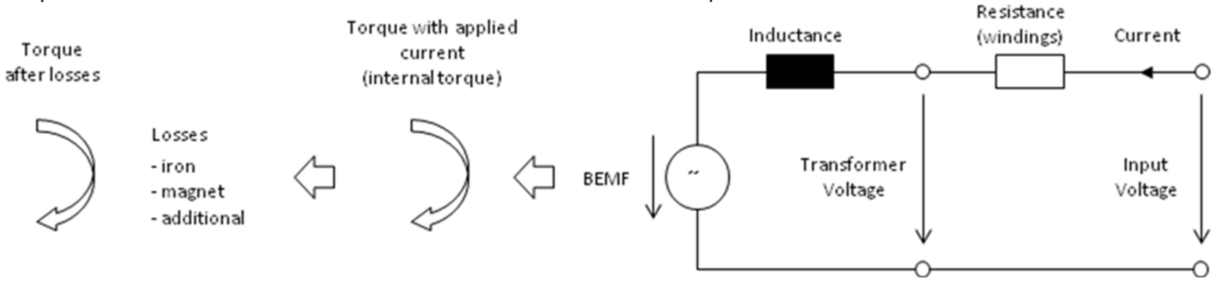
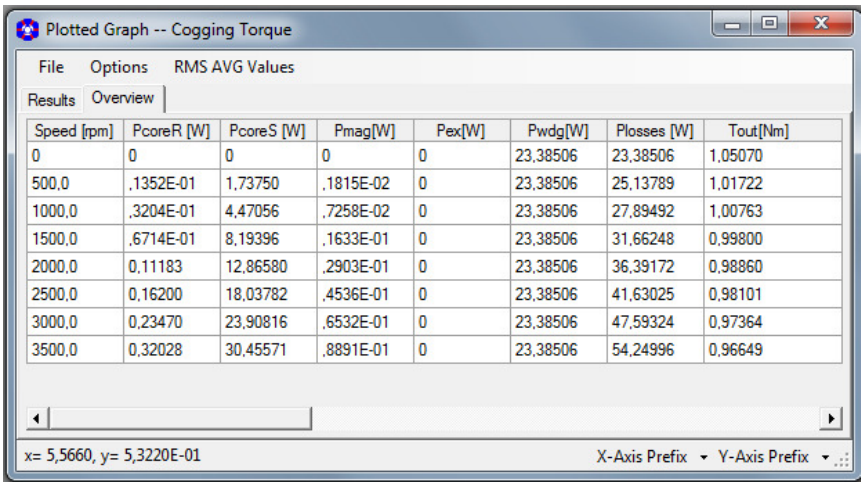


Fig. 79: Equivalent circuit diagram for synchronous motor

In *"Results"* an overview of the various result parameter (i.e. losses, internal /external torque, active power, apparent power, power factor, cosφ, efficiency, etc.) is displayed under the tab *"Overview".* Losses are only taken into account if they have been previously calculated.



Speed [rpm]	PcoreR [W]	PcoreS [W]	Pmag[W]	Pex[W]	Pwdg[W]	Plosses [W]	Tout[Nm]
0	0	0	0	0	23,38506	23,38506	1,05070
500,0	,1352E-01	1,73750	,1815E-02	0	23,38506	25,13789	1,01722
1000,0	,3204E-01	4,47056	,7258E-02	0	23,38506	27,89492	1,00763
1500,0	,6714E-01	8,19396	,1633E-01	0	23,38506	31,66248	0,99800
2000,0	0,11183	12,86580	,2903E-01	0	23,38506	36,39172	0,98860
2500,0	0,16200	18,03782	,4536E-01	0	23,38506	41,63025	0,98101
3000,0	0,23470	23,90816	,6532E-01	0	23,38506	47,59324	0,97364
3500,0	0,32028	30,45571	,8891E-01	0	23,38506	54,24996	0,96649

Fig. 80: Results parameter as "Overview" table

3.6.2 Cogging Torque and BEMF

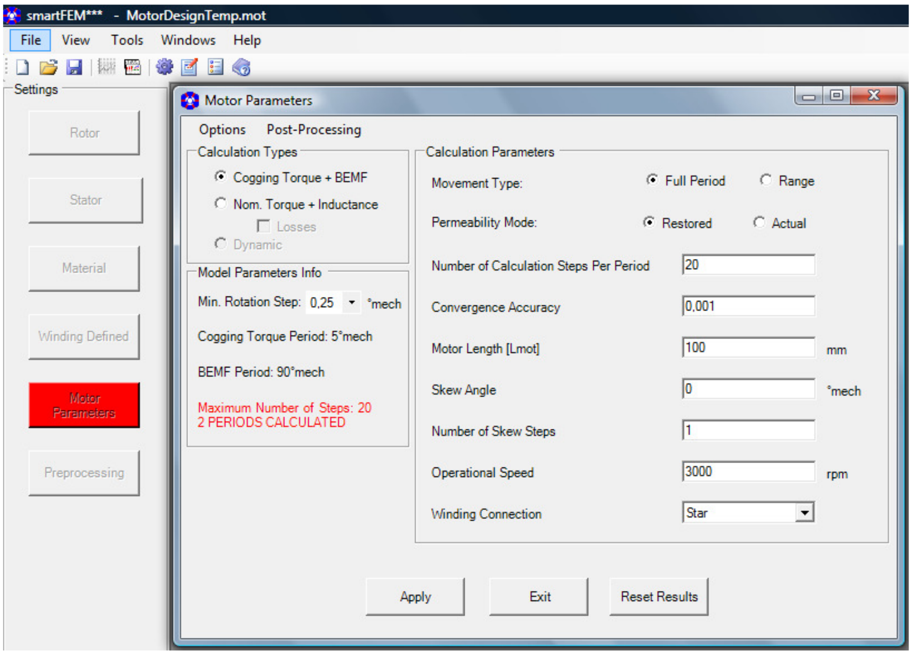


Fig. 81: Motor parameter - Cogging torque + BEMF

Model Parameters Info

- *Minimum Rotation Step x ° mech*
  Minimum angle of rotation of the rotor step in ° mech. This also corresponds to the node angle in mid air gap layer between rotor and stator. Through the selection box also different values can be selected. The adjustment should be set so, that the net elements in the central air gap layer become as quadratic as possible to ensure high accuracy of the power or torque calculations. When "smartFEM Settings - General" is set to manual entry of "Min. Rotation Steps ", any user-defined step width can be specified.
- *Cogging torque Period x ° mech*
  Cogging period in ° mech.
- *BEMF Period x ° mech*
  Period of the induced voltage in ° mech.
- *Maximum Number Of Steps x*
  Maximum number of calculation steps per period. 2 periods are calculated.

Calculation parameter

- *"Movement Type"*
  User can choose between the calculation for the entire period or a part of the period. When choosing a partial period, only cogging torque and flux are calculated.
- *"Permeability Mode"*
  With the help of this tab, the calculations in FEMAG are controlled. When "Restored" is selected, the calculations are started at each step angle with "zero". When "Actual" is selected, it starts with the result of the previous step angle. This has an influence on the number of iterations and, if necessary, also on the accuracy of the calculation results. This needs to be

evaluated and selected individually by the user. "Restored" ends in very good results, but it sometimes requires significantly more iterations and thus calculation time.

- *"Number of Calculation Steps per period"*
  The calculation of Cogging Torque + BEMF is carried out through two cogging torque periods. The specification of the minimum rotor rotational angle is dependent on the period length, in order to keep the calculation time within acceptable range that is justifiable to the accuracy of the curve.
- *"Convergation Accuracy"*
  Termination condition for the calculation accuracy of the permeability changes in FEMAG.
- *"Motor Length"*
  Motor length without projection = length of the axial airgap (see also chapter “*3.2.4 Relative Length”*).
- *"Skewing"*
  Herewith the effects of optimization are calculated by skewing of the rotor or stator. Total angle of inclination and the number of skew steps are indicated.

Skew Angle	0 = no inclination n = inclination in °mech over the entire motor length
Number Of Skew Steps	0 = no inclination 1 = continuous inclination on the specified angle n = number of laminated cores the first laminated core is always in the position of 0 ° mech!
Example:	Skew Angle = 15 °mech Number Of Steps = 4 Skew Angular offset of the laminated cores to each other = 15 °mech / (4-1) = 5 °mech

- *“Operational speed”*
  Rated speed of the motor in revolutions per minute.
- *“Winding Connection”*
  Winding circuit "star" or "triangle".
  With "Reset Results" previous calculated results are deleted, with "Apply" the entered data is accepted and the window is closed, with "Exit" the window is closed without saving the data.
  If the preprocessing has already been performed, then the calculation can be started by pressing the "Calculation" button.

„Apply, Exit, Reset Results“

With *“Reset Results”* previous calculated results are deleted, with *"Apply"* the entered data will be accepted and the window will be closed, with *"Exit",* the window is closed without changing any data which were entered before.

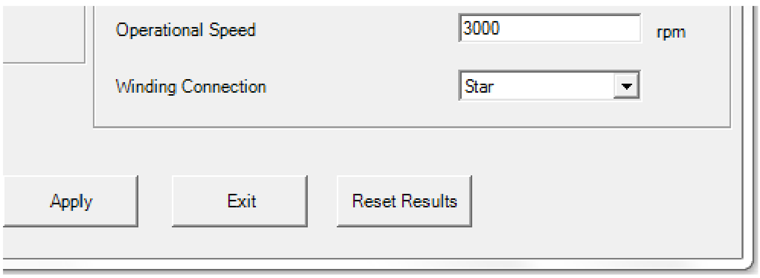


Fig. 82: "Apply, Exit" Motor Parameters

After “Preprocessing” has been performed before can now the calculations be started by pressing the button “Calculation”. The calculation results are shown in a graphic form. Both graphics and numerical result values can be copied to other applications via the clipboard.

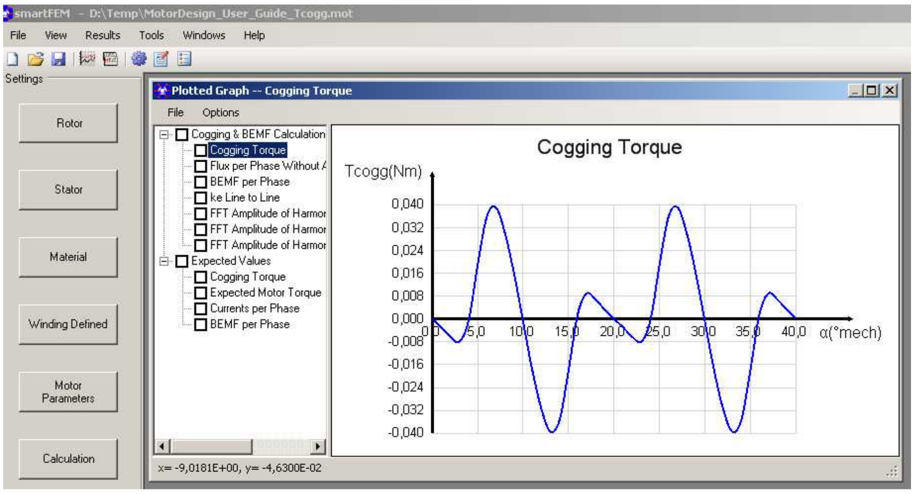
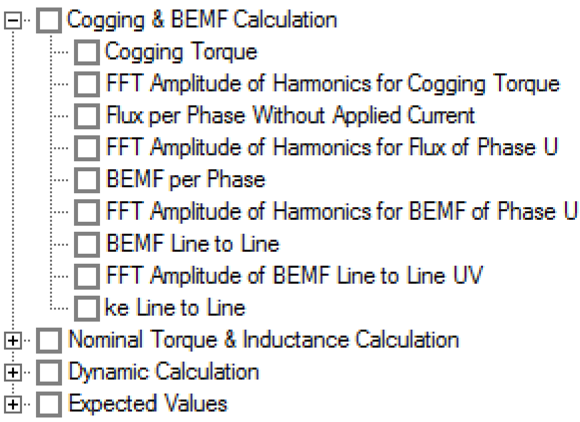


Fig. 83: Results - Results Cogging Torque

Overview of all the results of a "Cogging Torque + BEMF Calculation" in the form of graphs:



As smartFEM determines and stores the flux for each slot and rotor position, the results are determined without further FEM calculation if the winding scheme, motor length, skew angle, skew steps and speed are changed. This is displayed immediately after entering the changed parameters.

Fig. 84: Example of influencing the BEMF by inclination

3.6.3 Expected Values

In order to make a rapid prediction of the expected torque, the torque is determined based on the previously calculated BEMF using FEM and analytically calculated and displayed as "Expected Value".

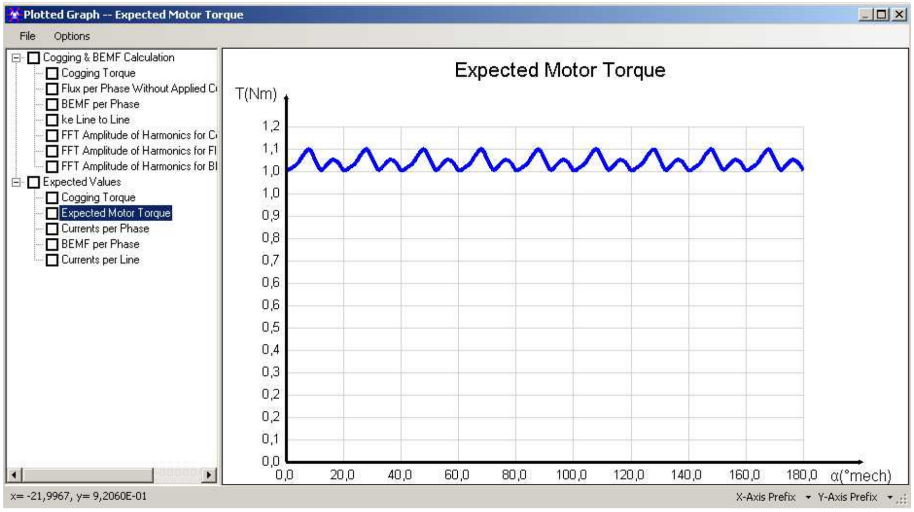
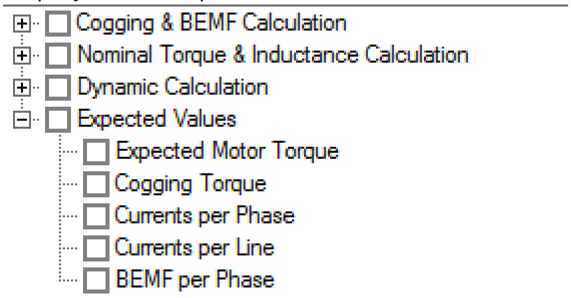


Fig. 85: Results - Expected Values

The calculation is performed according to the selection of constant phase current, sinusoidal phase current and trapezoidal or user-specific phase current. The selection can be seen in the menu „Nominal Torque + Inductance Calculation“ „Coil Excitation - Current Parameters“. Since no additional FEM calculation is necessary, the results are displayed immediately.

Overview of all the results of a "Cogging Torque + BEMF Calculation" in the form of a graph and displayed as "Expected Values":



3.6.4 Nominal Torque + Inductance

The calculation of the nominal torque is carried out in such a way that in each rotor position, the torque is calculated in de-energized and energized state using FEM. Subsequently, the nominal torque is calculated by difference formation. The adjustment of calculation parameters can be performed in *"Motor Parameters".*

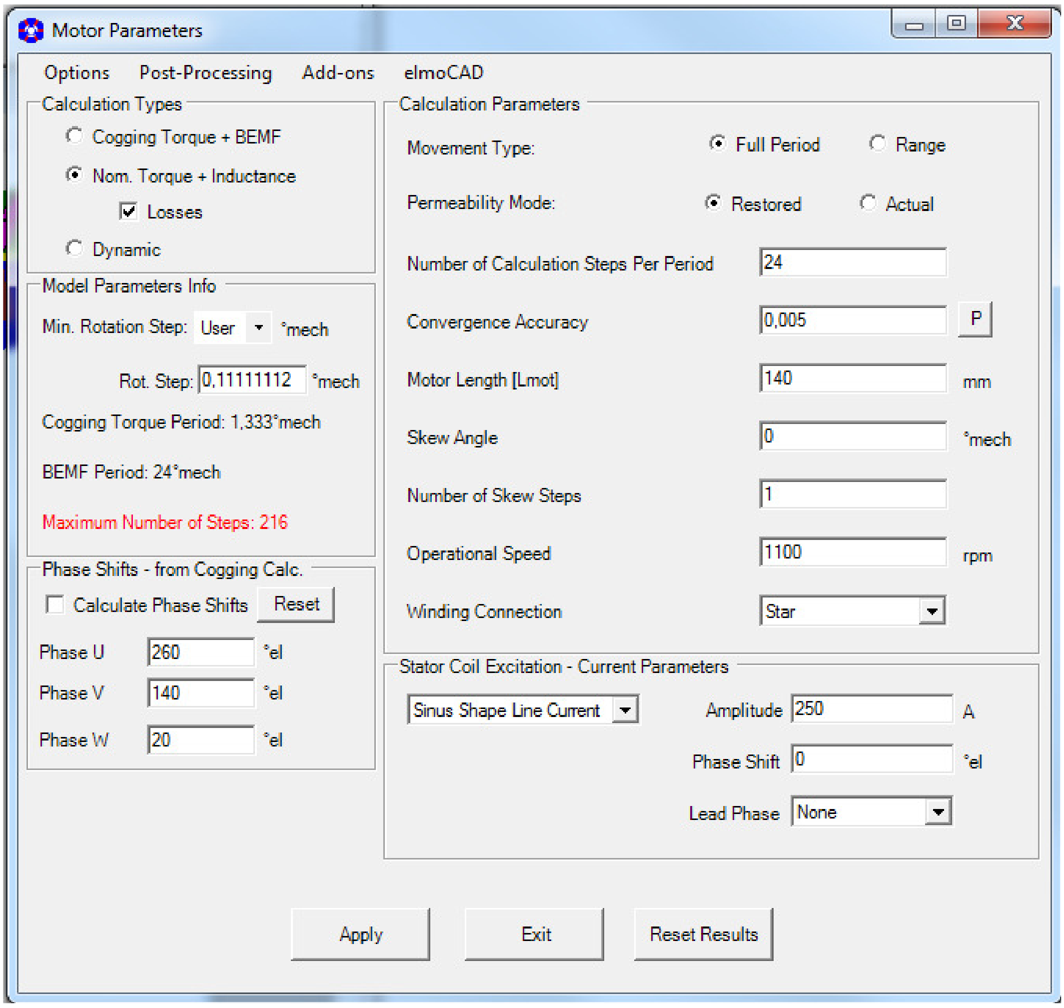


Fig. 86: Motor Parameters - Nominal Torque + Inductance

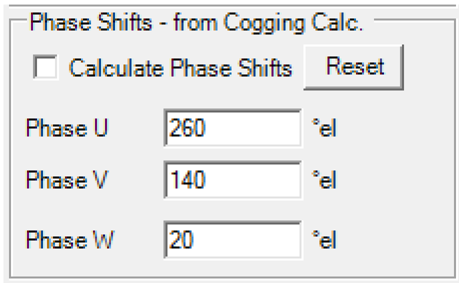
3.6.4.1 Model Parameters Info

- *Minimum Rotation Step x ° mech*
  Minimum angle of rotation of the rotor step in ° mech. (according to 3.6.2Cogging Torque and BEMF)
- *Cogging Torque Period x ° mech*
  Cogging period in ° mech.
- *BEMF Period x ° mech*
  Period of the induced voltage in ° mech.
- *Maximum Number Of Steps x*
  Maximum number of calculation steps.

3.6.4.2 Phase Shifts

- **from Cogging Torque + BEMF Calculation**
- **from Nominal Torque + Inductance Calculation**

- By execution of *"Cogging Torque + BEMF Calculation"* the phase angles between the individual windings are determined, which is then displayed in *"Nominal Torque + Inductance Calculation"*.



- If „Nominal „Torque + Inductance Calculation“ is selected, the phase angles are displayed if previously „Cogging Torque + BEMF Calculation“ was performed.
- If no „Cogging Torque + BEMF Calculation“ was carried out, „Phase Shifts: Not Defined“ is displayed.

  If the check sign is set on "Calculate Phase Shifts“, the phase positions are automatically recalculated before the nominal torque calculation.

  For this purpose a torque calculation is performed with constant current for each phase. The angles (° el) of the maxima of the torques correspond to the positions of the respective phase. Then the actual *"Nominal Torque + Inductance"* calculation is carried out.
- „Phase U/V/W: not defined“ is displayed for the motor types „Switched Reluctance“ and „Universal Motor“. Since these types of motors do not include magnets, it is not possible to calculate BEMF and the phases of the windings. These are then determined before performing the actual torque calculation as previously described. (See also 6.3 Switched Reluctance Motor)

3.6.4.3 Calculation Parameters

- *"Movement Type"*
  User can choose between the calculation for the entire period or a part of the period.
- *"Permeability Mode"*
  This is used to control the calculations in FEMAG. When *"Restored"* is selected, the calculations are started at each step angle with *"zero"*. When *"Actual"* is selected, it starts with the result of the previous step angle. This has an influence on the number of iterations and, if necessary, also on the accuracy of the calculation results. This needs to be evaluated and selected individually by the user. "Restored" ends in very good results, but it sometimes requires significantly more iterations and thus calculation time.
- *"Number of Calculation Steps per period"*
  The calculation of Cogging Torque + BEMF is carried out through two cogging torque periods. The specification of the minimum rotor rotational angle is dependent on the period length, in order to keep the calculation time within acceptable range that is justifiable to the accuracy of the curve.
- *"Convergation Accuracy"*
  Termination condition for the calculation accuracy of the permeability changes in FEMAG.
- *"Motor Length"*
  Motor length without overhangs.
- *"Skewing"*
  Herewith the effects of optimization are calculated by skewing of rotor or stator. Total angle of inclination and the number of skew steps are indicated.

Skew Angle	0 =	no inclination
	n =	inclination in ° mech over the entire motor length
Number Of Skew Steps	0 =	no inclination
	1 =	continuous inclination on the specified angle
	n =	number of laminated cores,

the first laminated core is always in the position of 0 ° mech!

Example:

Skew Angle = 15 °mech
Number Of Steps = 4 Skew
Angular offset of the laminated cores to each other = 15 °mech / (4-1) = 5 °mech

Note: In the current smartFEM version, the nominal torque calculation and the loss calculation are only carried out for Skew Angle = 0 °!

- *“Operational speed”*
  Rated speed of the motor in revolutions per minute.
- *“Winding Connection”*
  Winding circuit *"Star"* or *"Delta"*

3.6.4.3.1 Skewing of Rotor and Stator

Skewing of rotor poles or stator teeth by a pole pitch is an appropriate possibility to reduce cogging torque and harmonics of the BEMF.

Besides continuous skewing is it also possible to assemble the rotor with several slices in axial direction (discrete skewing):

- The single slices are skewed to each other by an angle = skew angle / number of slices
- Skewing with an infinite number of slices is then conform to continuous skewing

The impact of skewing on cogging torque, etc. can easily be determined by superposition using an Excel sheet.

In smartFEM are the results immediately displayed after entering of skew angle and number of slices (number of steps). For continuous skewing has number of step = 1 to be entered.

Definition:

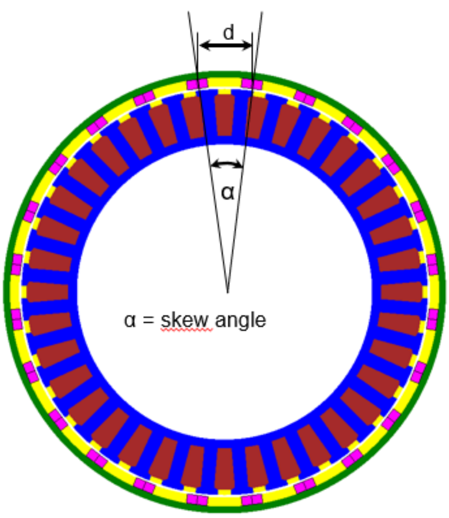
$d = \alpha$ [°mech] / 360° * 2 * Pi * r_{magnet}

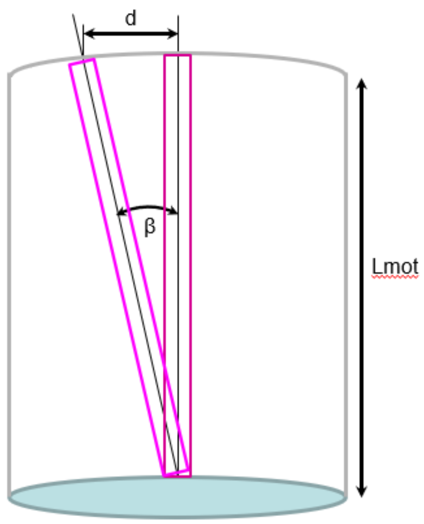
= Ddistance of the center points of the magnet areas at front and rear side.

Example:
d = 5°/360*2*Pi* 71,06mm = 6,2mm

β [°] = d / (2 * Pi * Lmot) * 360°

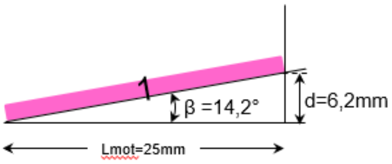
Example:
β = 6,2/(2*Pi*25)*360 = 14,2°



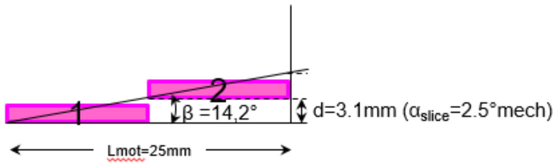


Example with α = 5 °mech

Continuous skewing



discrete skewing with 2 rotor slices



Rotor Positions & Skewing

Skew Angle = α [°mech]

Number of Skew Steps = Number of slices (NoSteps)

Step Width [°mech] = Skew Angle / Number of Skew Steps

Example mit α = 5 °mech

NoSteps	Step Width	Position per Step [°mech] = skew angle per slice			
2	α /2 = 2,5°mech/step,	1.step α=0°,	2.step α=2.5°		
4	α /4 = 1,25°mech/step,	1.step α=0°,	2.step α=1.25°,	3.step α=2,5°,	4.step α=3,75°
8	α /8 = 0,625°mech/step,	1.step α=0°,	2.step α=0,625°,	3.step α=1,25°, … ,	8.step α=4,375°
16	α /16 = 0.3125°mech/step,	1.step α=0°,	2.step α=0,3125°,	3.step α=0,625°, … ,	16.step α=4,6875°

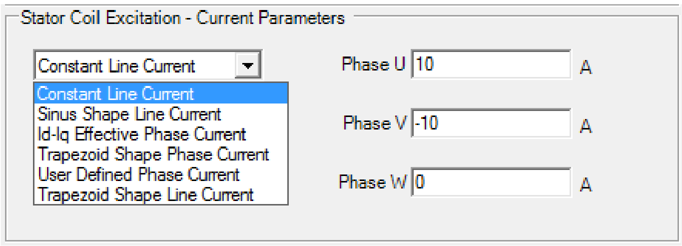
…

NoSteps >>16steps → continuous skewing, equivalent NoSteps = 1 in smartFEM

3.6.4.4 Stator Coil Excitation - Current Parameters

Different current waveforms can be entered:

- Constant current with amplitude indication



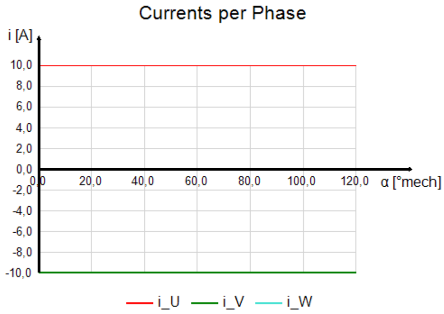
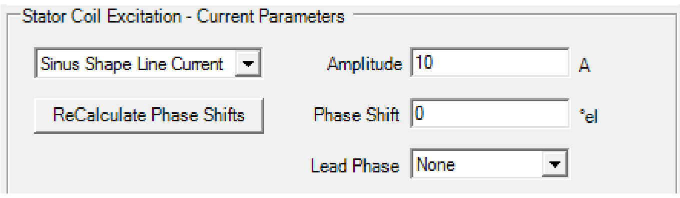


Fig. 87: Stator Coil Examination - Constant Line Current

If the "Calculate Phase Shifts" button is pressed, the positions of all phases are determined before the actual nominal torque calculation.

- Sinusoidal current with amplitude indication and phase shift between current and BEMF per line



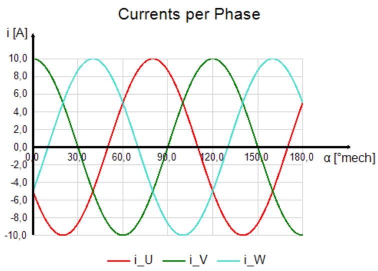
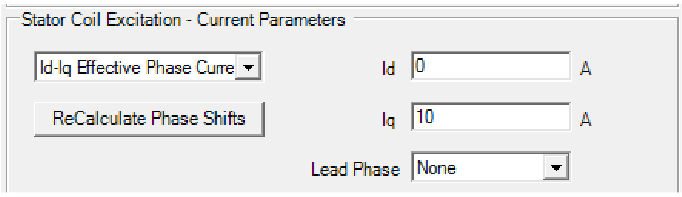


Fig. 88: Stator Coil Examination - Sinus Shape Line Current

- Id-Iq with indication of the currents in the d- and q-axis:



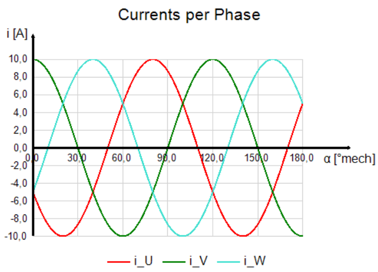
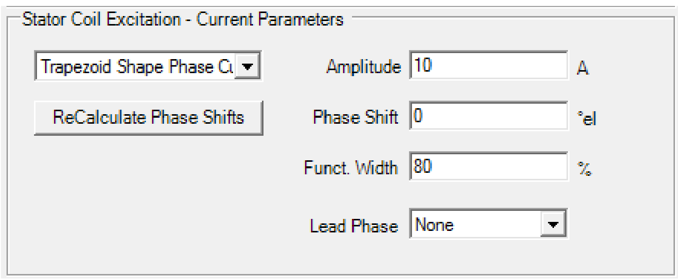


Fig. 89: Stator Coil Excitation - Id-Iq Effective Phase Current

- trapezoidal curve shape of the phase currents with amplitude indication, phase shift and function width. The function width is the range in which the amplitude is 100%. Flanks with a linear ascent and descent are shown in between:



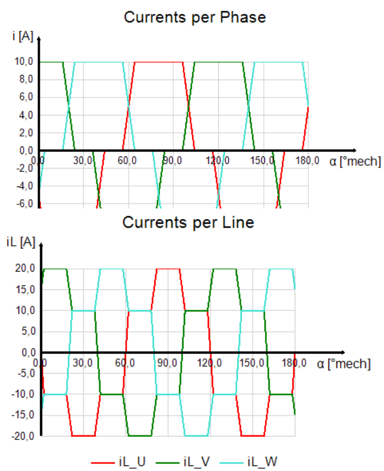
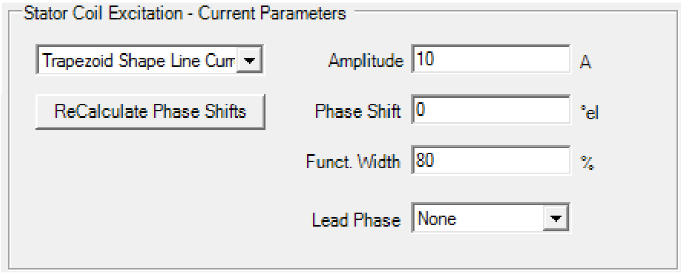


Fig. 90: Stator Coil Excitation - Trapezoid Shape Phase Current



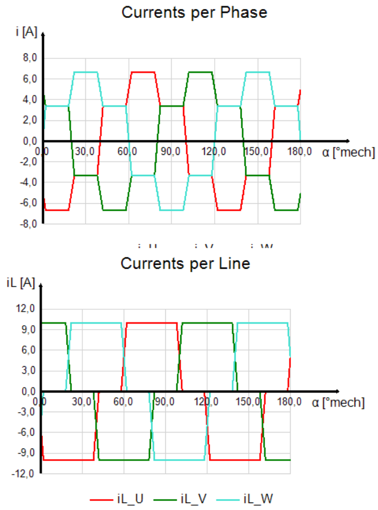


Fig. 91: Stator Coil Excitation - Trapezoid Shape Line Current

- *"Lead Phase"*

  In case that non-symmetrical phase currents or phase positions of the BEMF result from geometry and/or winding scheme (which have a phase shift unequal to 360°/total number of phases), a single phase can be set as leading. The currents of other phases are then automatically defined in such a way that an equal phase shift occurs between all the currents. This is necessary to simulate the behavior of conventional controls that cannot produce unsymmetrical phase shifts.

  In case where no leading phase is selected, the phase shifts between the phase currents are defined when their result from the unsymmetrical BEMF.

- user-specific curve shapes of the phase currents:

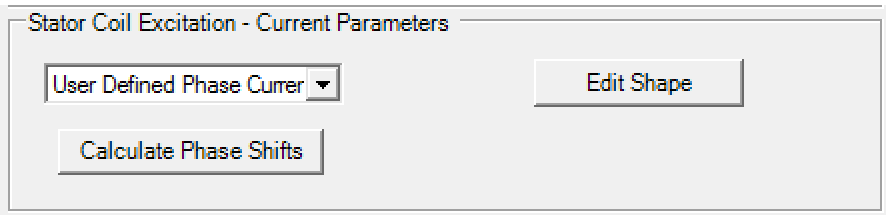


Fig. 92: Stator Coil Excitation - User Defined Phase Current

The corresponding settings can be made via the *"Edit Shape"* button. A table with a sinusoidal current profile is created as default.

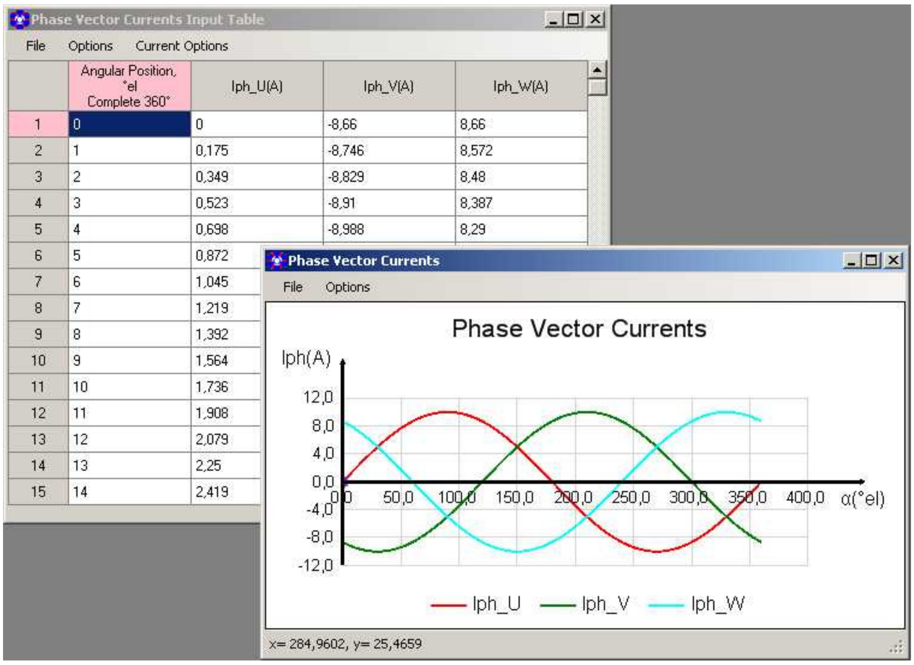


Fig. 93: Stator Coil Excitation - Phase Vector Currents Input Table

The user can directly modify the values contained in the table, make settings via *"Options"* or copy and paste a table of values created in another tool.

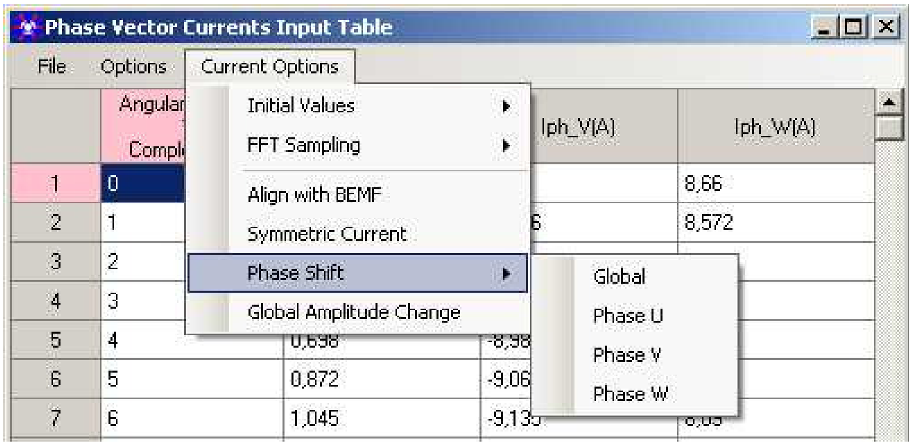


Fig. 94: Phase Vector Currents Input Table - Current Options

With *"Current Options"* the following settings are possible:

"Initial Values":	*Save or load the saved value table.*
"FFT Sampling":	*Sampling rate for Fourier decomposition, default = 1000*
"Align with BEMF":	*Adjustment of the phase positions of the current waveforms according to the BEMF per phase*
"Symmetric Current":	*Current values input in phase U are copied to the other phases with a phase shift of 120 ° and 240 °.*
"Phase Shift":	*A phase shift can be entered individually for all or each phase.*
"Global Amplitude Change":	*The amplitudes are multiplied by the entered factor.*

„Apply, Exit, Reset Results“

With *“Reset Results”* previous calculated results are deleted, with *"Apply"* the entered data will be accepted and the window will be closed, with *"Exit",* the window is closed without changing any data which were entered before.

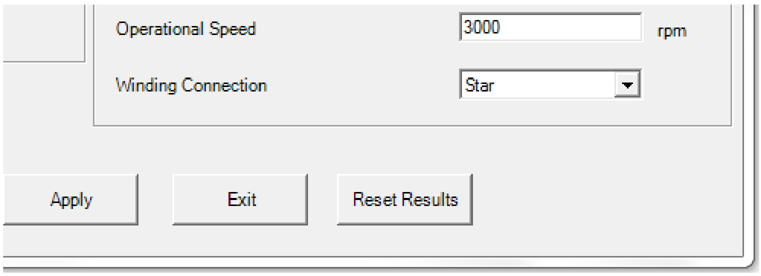


Fig. 95: Motor Parameters - "Apply, Exit, Reset Results"

3.6.5 Calculation

If the *"Preprocessing"* has already been carried out the calculations can now be started by pressing the button „*Calculation*". After calculations are finished, can the results be presented in different graphs.

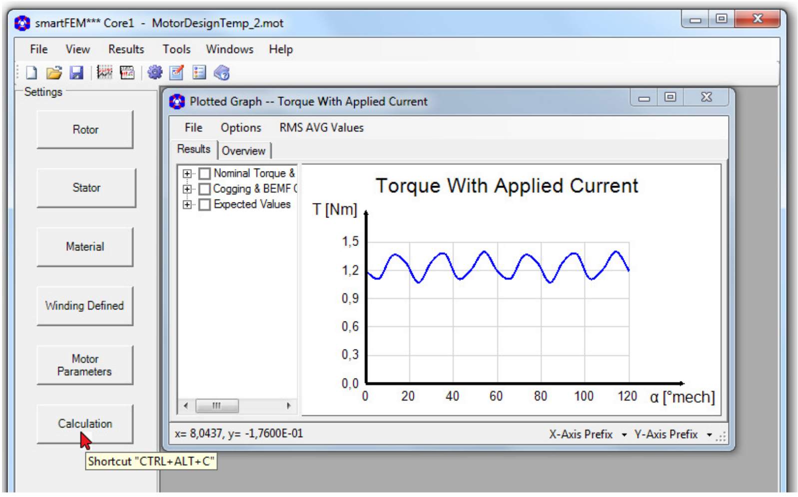


Fig. 96: Display of results for nominal torque with sinusoidal current

Overview of all the results that can be presented as a graph after performing *"Nominal Torque + Inductance Calculation":*

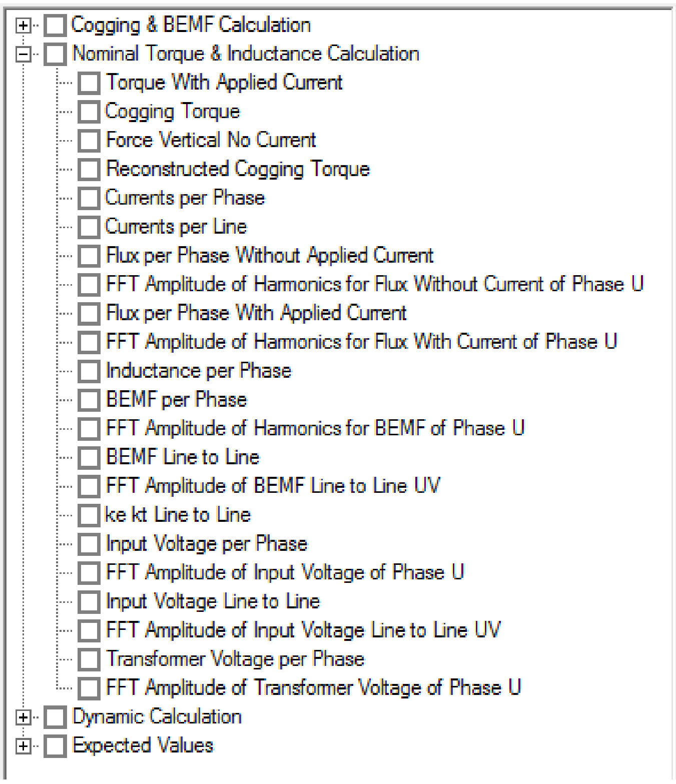


Fig. 97: Overview Calculation Results

If afterwards skewing parameters or motor length are changed, all forces and torques are also recalculated without the need for new FEM calculations.

3.6.6 Dynamic calculations

Dynamic calculations can be carried out for motor start-up and steady-state operation. The calculations are carried out analytically in the time step method on the basis of equivalent circuit diagrams, whereby the inductances can also be determined in co-simulation with FEMAG.

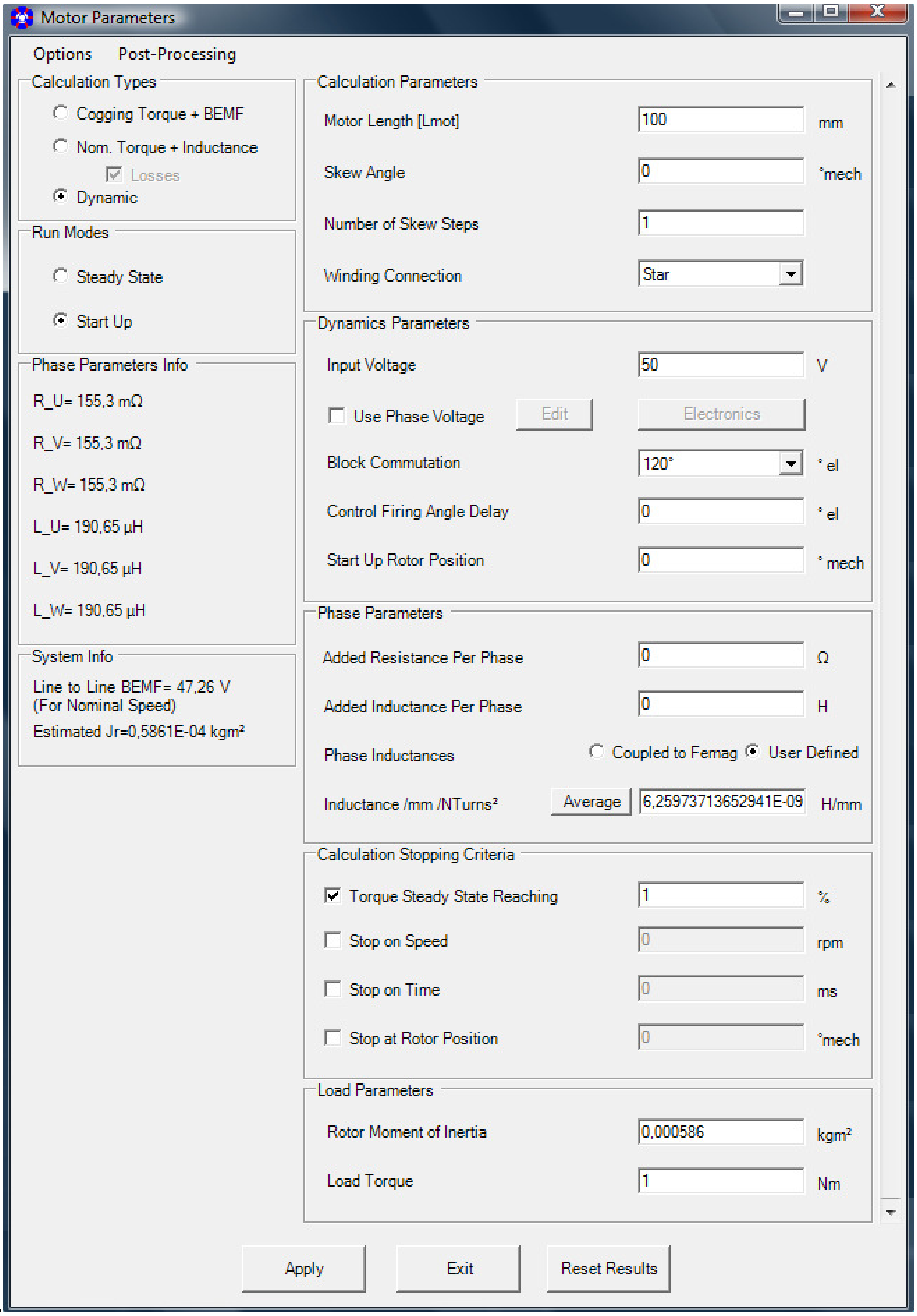


Fig. 98: Motor parameters for dynamic calculation of stationary operating conditions

“Run Modes”

- „Steady State“ for steady state operation
- "Start Up" for the motor start.

“Phase Parameters Info”

- Resistors RU, RV, RW and inductors LV, LU, LW of all windings which belong to a phase
- Induced terminal voltage
- Moment of inertia of the rotor

“System Info”

- Line to Line BEMF (for Nominal Speed)
- Estimated Jr

“Calculation Parameters”

- "Motor Length"
  Motor length without overhangs
- "Skewing"

  Herewith the effects of optimization are calculated by skewing of rotor or stator. Total angle of inclination and the number of skew steps are indicated.

  “Skew Angle” 0 = no inclination
  n = inclination in ° mech over the entire motor length

  “Number Of Skew Steps” *0 = no inclination*
  1 = continuous inclination on the specified angle
  n = number of laminated cores,
  the first laminated core is always in the position of 0 ° mech!

  Example:
  Skew Angle = 15 °mech
  Number Of Steps = 4
  Angular offset of the laminated cores to each other =
  15 °mech / (4-1) = 5 °mech

- "Winding Connection"
  Winding circuit "Star" or "Delta"

“Dynamics Parameters”

- "Input Voltage"
  Maximum value of an applied constant phase voltage
- Use Phase Voltage
  Instead of the constant phase voltage, through "Input Voltage", user-specific phase voltages can also be used. In this case, the phase characteristics, the amplitude and the phase angle can be set as needed. This function can currently only be used in "Run Mode" and "Steady State".

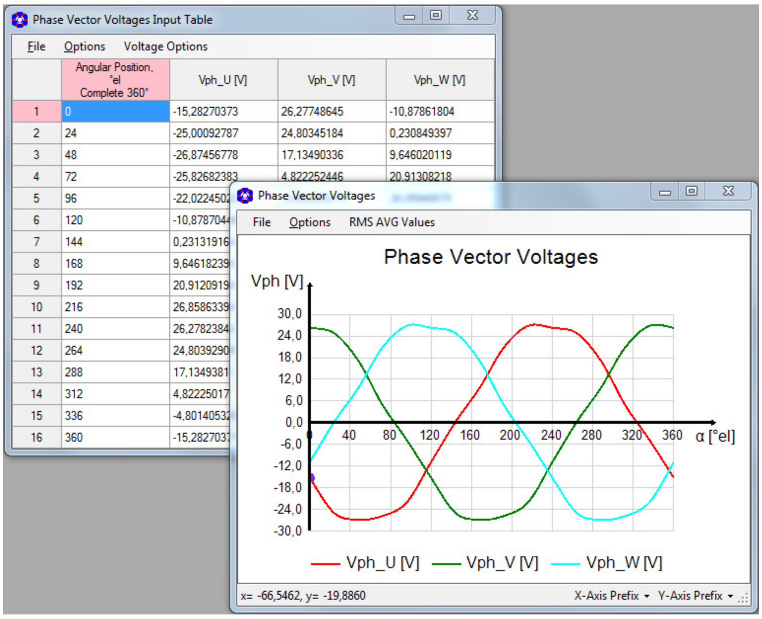


Fig. 99: Use of user-specific phase voltages

With *"Voltage Options"* can user perform various functions to adjust voltages:

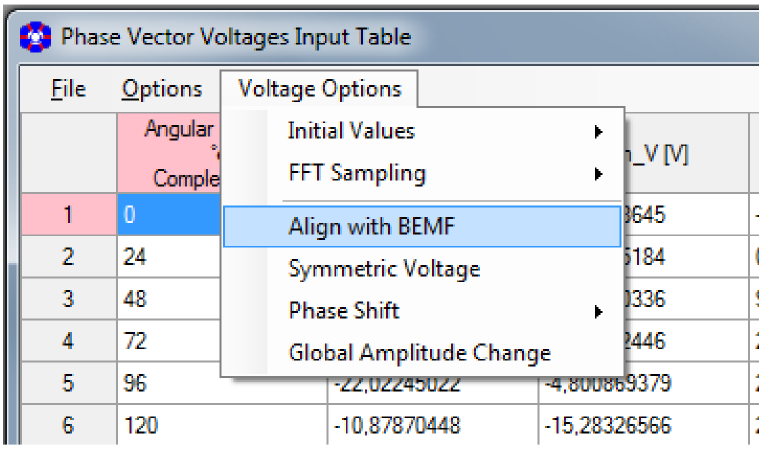


Fig. 100: Voltage Options

- *"Initial Values"*: Save, Restore
- *"FFT sampling"*: Input of a value, Default = 1.000
- *"Align with BEMF"*: Set to the same phase angle as BEMF
- *"Symmetric Voltage"*: Setting the phases V and W as U with offset of 120 °el
- *"Phase Shift"*: Input of a phase angle for all phases equal or different, per phase
- *"Global Amplitude Change"*: Input of a multiplication factor to change the peak value of the voltage.

"Electronics" is used to specify the resistances of the transistors and diodes of the inverter in the equivalent circuit diagram and the time steps of the calculations. Thus, with a good approximation, any PWM signal can be used for the input voltage of the motor in 3-phase star connection.

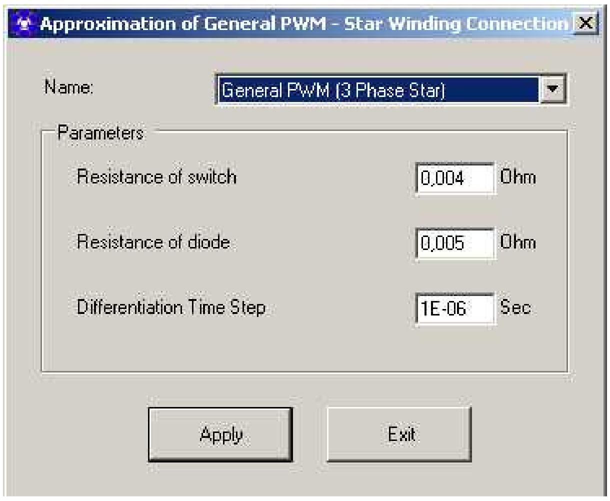


Fig. 101: Setting the parameters for the inverter model

- *"Block Commutation"* Input of electrical angle of the commutation duration 120°el or 180°el.
- *"Control Firing Angle Delay"* Delay angle of commutation time.
- *"Stop Calculation Criteria"* Abort conditions for completing the calculations are linked with "or" and can be selected and parameterized.

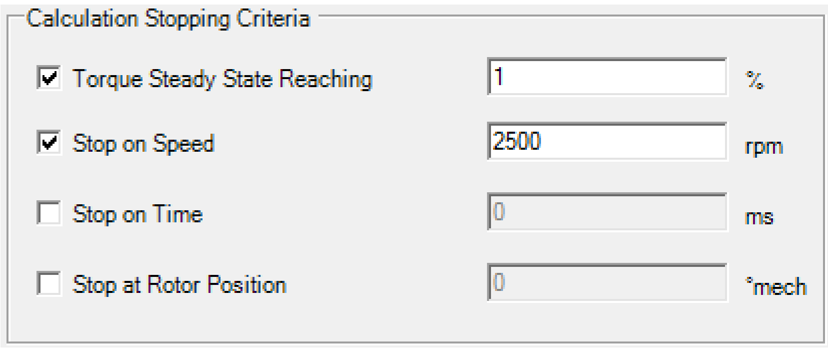


Fig. 102: Termination conditions for dynamic calculations

Phase Parameters

- "Added Resistance per Phase" User-specific resistance added as a constant to the phase resistance.
- "Added Inductance per Phase" User-specific inductance added as a constant to the phase inductance.
- *"Phase Inductances"*
  - *"Coupled to FEMAG"* The inductances are determined for every rotor position by using co-simulation by FEMAG.
  - *"User Defined"* The inductance is calculated from the winding scheme independently of the rotor position and is specified as a constant value.

The calculation results are displayed as graphs. Both graphics and the numerical result values can be copied to other applications via the buffer (see also 0).

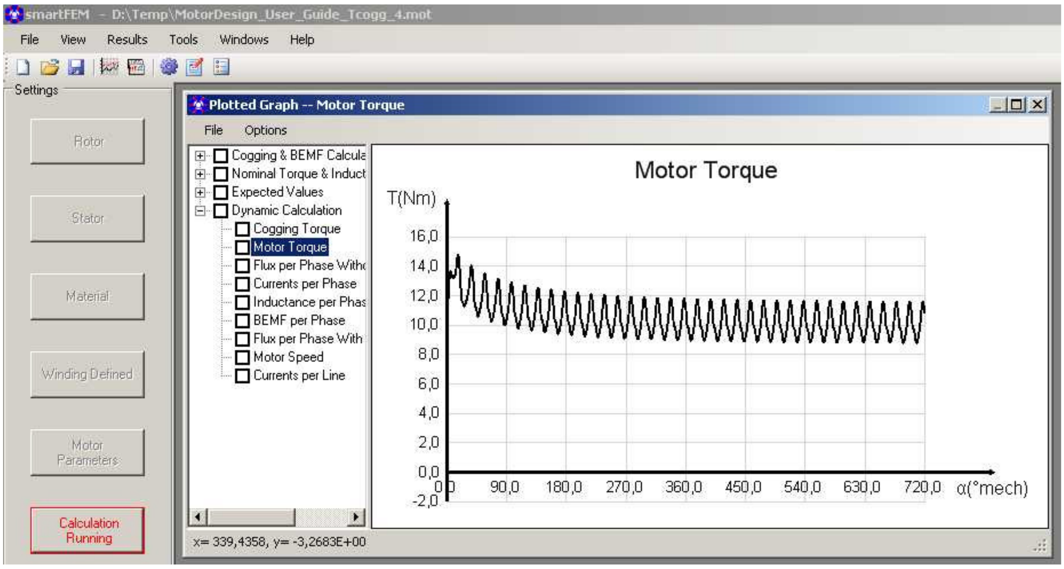
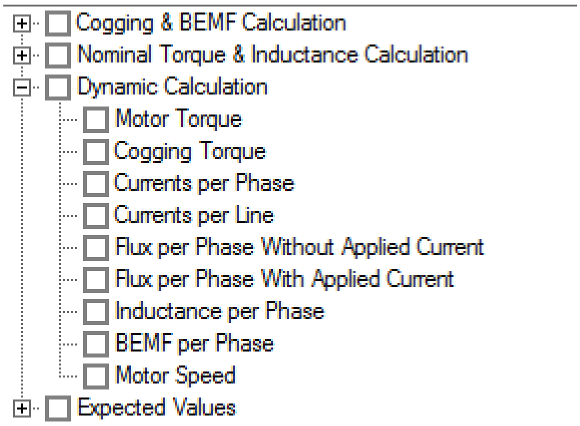


Fig. 103: Dynamic calculation of the torque curve versus rotor position

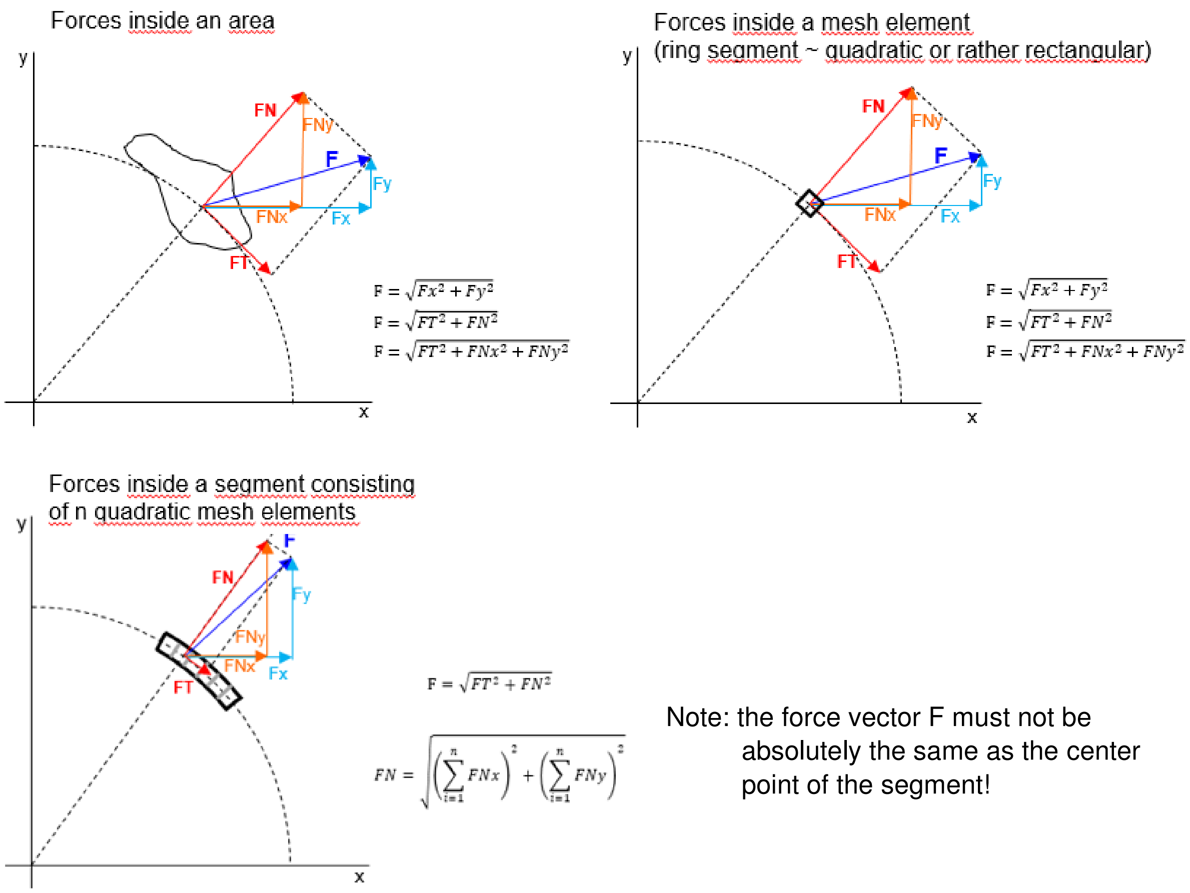
Overview of all the results that can be displayed as graphs after performing "Dynamic Calculation":



3.6.7 Postprocessing - Forces and Torque

3.6.7.1 Force Calculation

Basics



F = Force Vector
FT = tangential component of the force vector
FN = normal (radiale component of the force vector
FNx = x-component of the normal force vector
FNy = y-component of the normal force vector

Fig. 104: Basics of Force Vectors

smartFEM offers the calculation of the tangential and radial forces inside of the middle airgap layer.

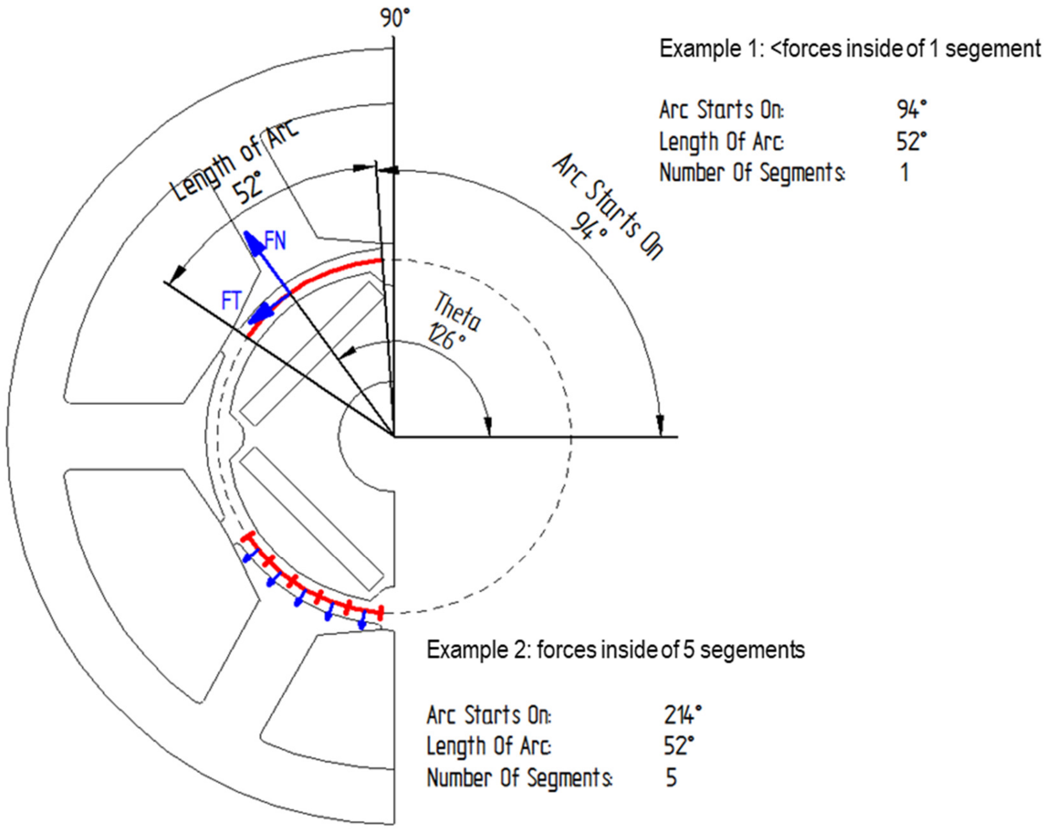


Fig. 105: Force Calculation with smartFEM

Example: results of force calculations of a motor with centric and eccentric rotor

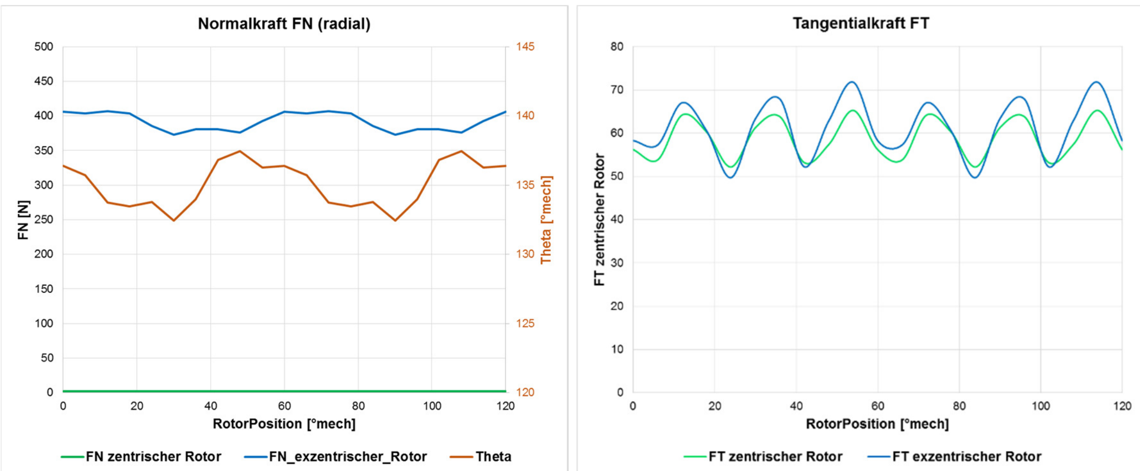


Fig. 106: Example results of force calculations

For input of parameter and output of results can the window "*Force Calculation*" be opened via the window "*Motor Parameters*" and menu "*Postprocessing*". Following parameter can be entered:

- *„Arc Starts on"*
  Start of the angle of the force calculations
- *„Length of Arc"*
  Length of the arc inside which the forces are calculated

  All angles are specified as polar-coordinates
- *„Number of Segments"*
  Number of the arc segments (each with the same length) where inside each the normal and tangential forces are calculated.
- Selection *„Cogging Torque"* or *„Nominal Torque"*
  Selection which results of field calculations will be used for the force calculations. If „Nominal Torque" is selected then are the defined currents considered.

The results are displayed in a table and can be copied by the menu *„Options"*→*„Copy Results"* via the clipboard into other software programs.

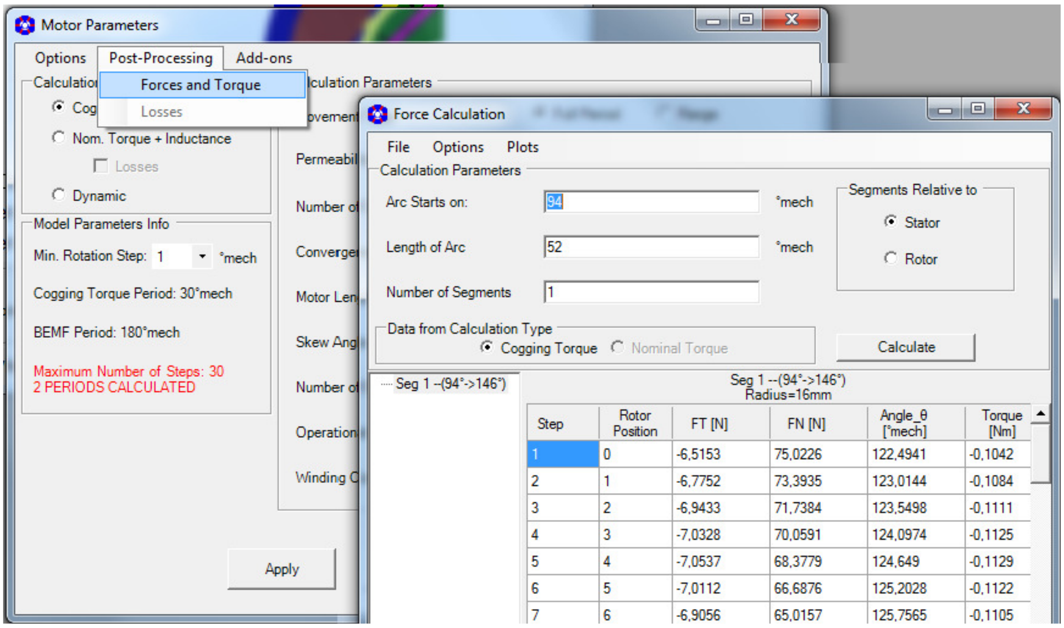


Fig. 107: Example 1 - force calculation in one segment

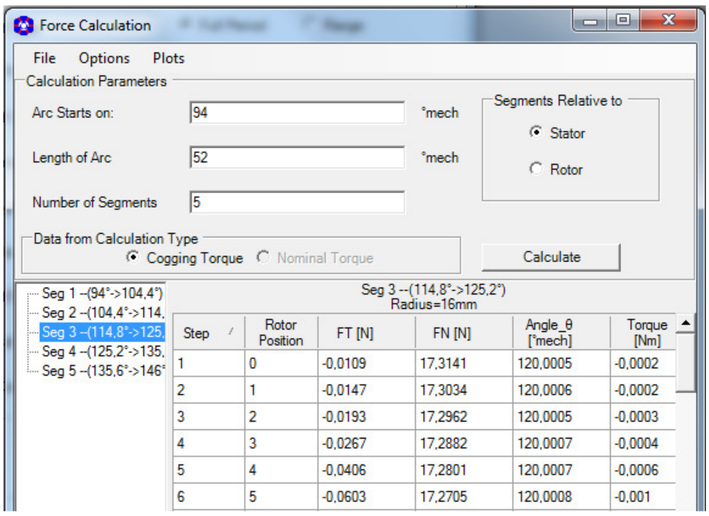


Fig. 108: Example 2 - force calculation in five segments

3.6.7.2 Torque

The calculation results of the torque in smartFEM main menu “*Results*“ are rounded to three significant digits by the FEM solver FEMAG. This leads not to accurate results when small torque ripples should be evaluated. Therefore the function „*Postprocessing - Forces and Torque - Force Calculation*“ with higher number of significant digits of the results is used whereby the force calculation has to be performed for one segment by “*Arc Starts on*“ = 0° and “*Length of the Arc*“ = 360°.

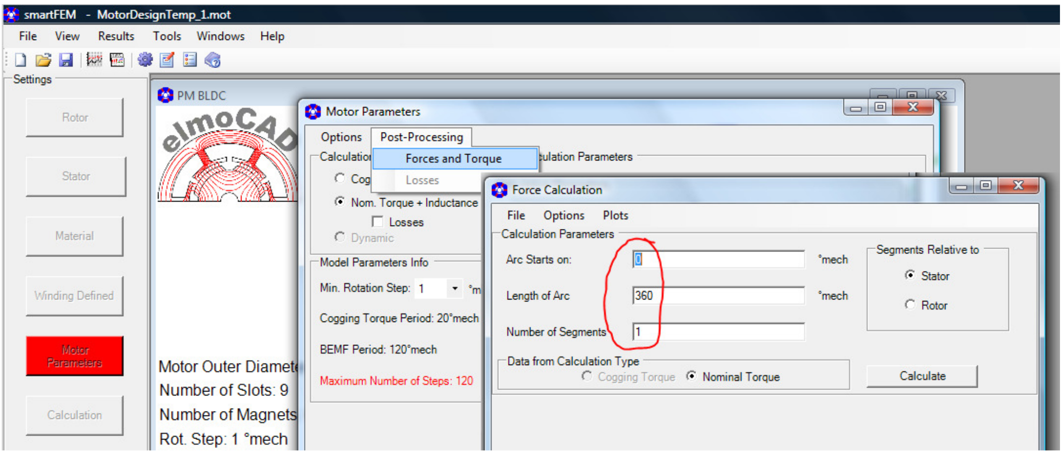


Fig. 109: Postprocessing - Forces and Torque

Afterwards can plots of the torque and its harmonic be displayed and evaluated.

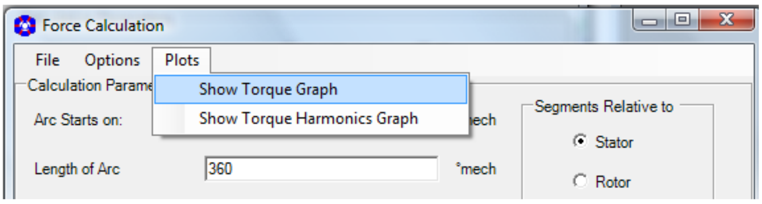


Fig. 110: Postprocessing - show torque graphs

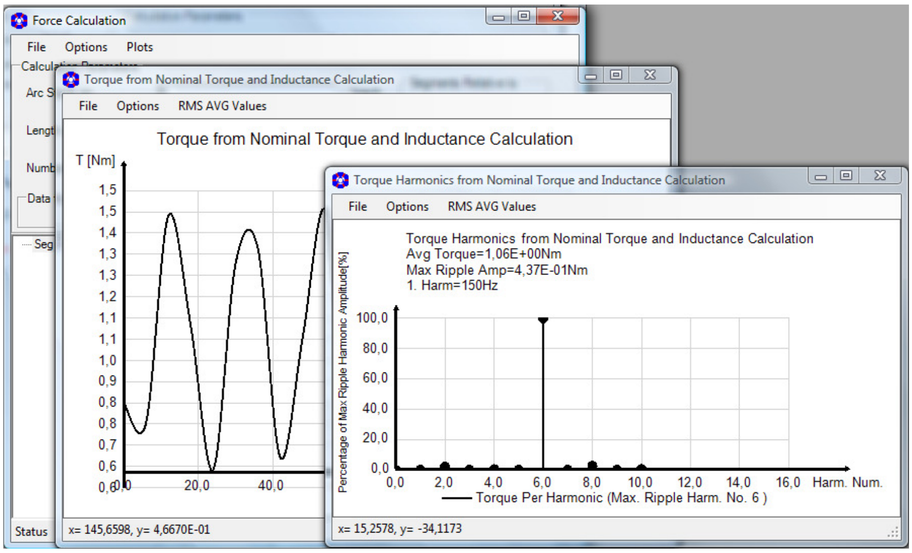


Fig. 111: Postprocessing - Plots of torque and its harmonic

Via the menu „*Options - Copy Results*“ can the results values of the torque and the flux density in the airgap be copied to a spreadsheet analysis for special user defined evaluations.

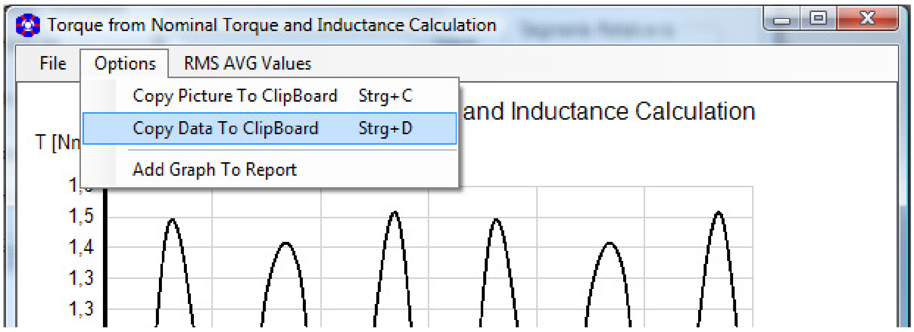


Fig. 112: Postprocessing - selection of data to copy the results of the torque

3.6.8 Post Processing - Loss Calculation

With smartFEM can copper, iron and magnet losses be calculated in the rotor and the stator. The necessary measurement and saving of basic data is accompanied by the *"Nominal Torque + Inductance"* calculation provided that beforehand the selection *"Losses"* in *"Motor Parameters"* was chosen.

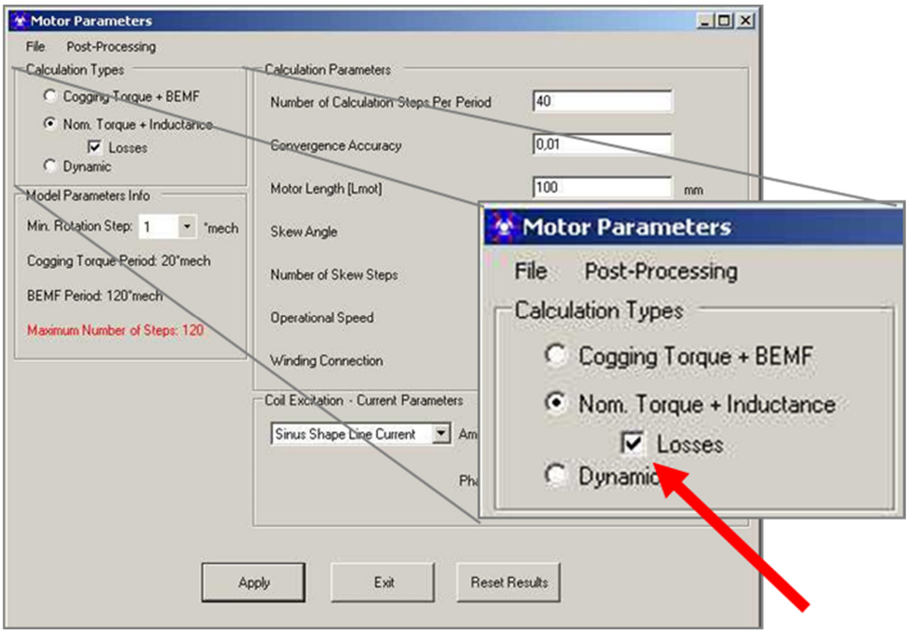


Fig. 113: Motor Parameters - Selection field "Losses"

By selecting the parameter *"Losses"*, all inductions for all network elements for each rotor position are stored. This results in an increased memory footprint for the motor model to follow. Thanks to this, the user can then calculate the losses at different speeds and the use of different materials very quickly, without starting the time consuming "Nominal Torque + Inductance" running calculation.

After performing the calculations for *"Nominal Torque + Inductance"* the losses can be calculated through the *"Postprocessing"* menu.

By default, in addition to this, the material data selected in the *"Materials"* menu are displayed in the window *"LossCalculation"*, which serve as the basis for all FEM calculations. Other materials can also be selected. However, the loss calculation using these materials serves only for approximate evaluation because the values of induction from FEM calculations with the materials selected in *"Materials"* are used. The same applies to the conductivity of the magnets in *"Magnets Conductivity"*.

For information, the values of the BH curve appear when the mouse points on the ⓘ sign.

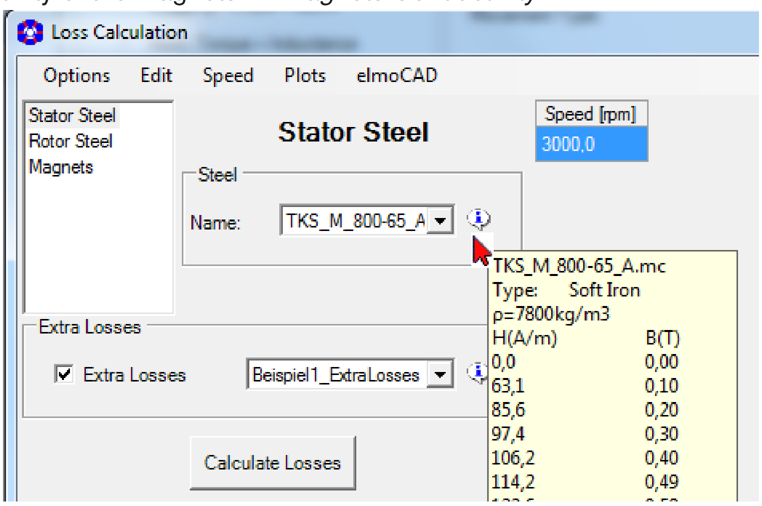


Fig. 114: Loss Calculation - Parameters

Via *"Speed - Add Speed Range"* further speeds for the loss calculation can be specified.

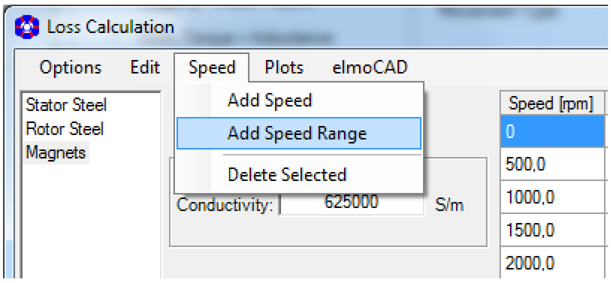


Fig. 115: Loss Calculation - Add Speed Range

In addition, the number of magnetic segments in the axial z-direction can be assigned, which reduces the magnet losses caused by eddy currents.

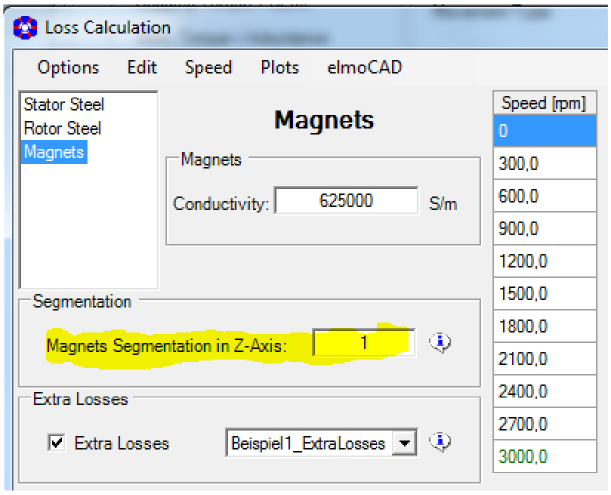


Fig. 116: Loss Calculation - Magnets Segmentation

By selection of *"Extra Losses"* a file *.pex with additional losses (for example, friction, fans, etc.) can be selected in case that this file was created as text file *.txt and saved in the material folder *„...\smartFEM\Materials"*. The separation of the columns must be the *"Tab"* sign.

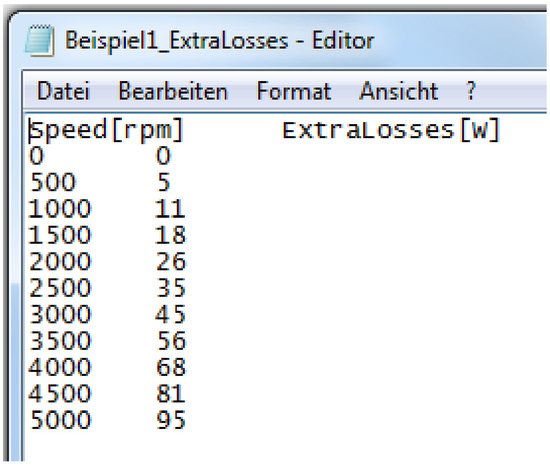
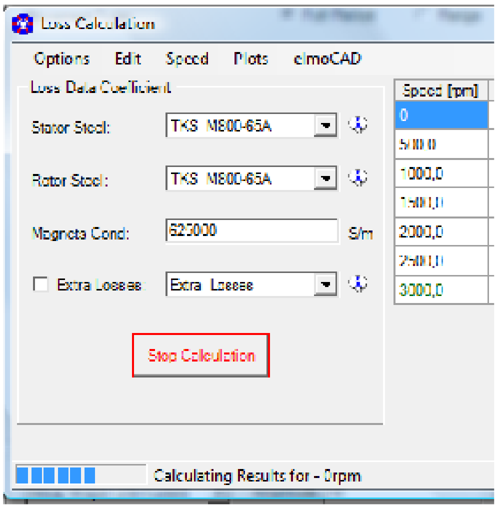


Fig. 117: Loss Calculation - Example Textfile.pex



After pressing the *"Calculate Losses"* button, the calculations are performed. During this time the *"Calculating"* button appears.

A running calculation is aborted by pressing the *"Calculating"* button.

Fig. 118: Loss Calculation - Calculation Process

The results of the loss calculation are displayed immediately after the calculation in tabular form and in plots:

- Iron losses in the rotor
- Iron losses in the stator
- Magnetic losses
- Additional losses
- Copper losses in the windings
- Plots with the loss densities
- Plots with the harmonics of rotor and stator losses

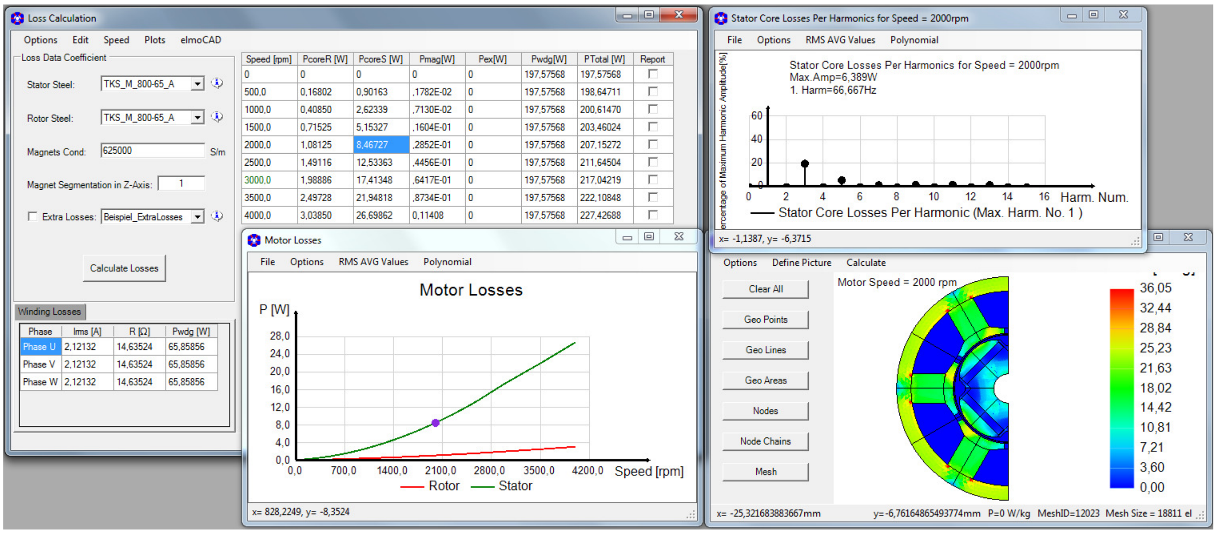
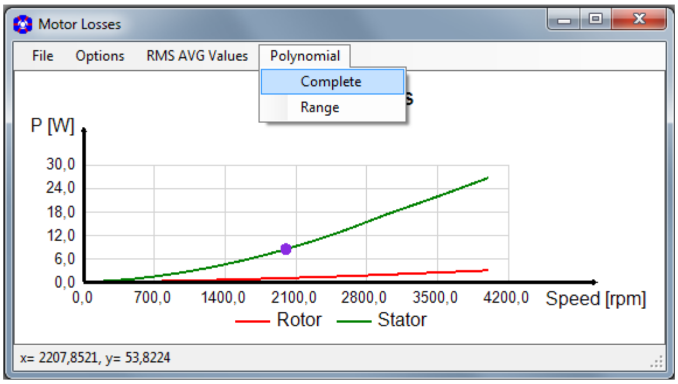
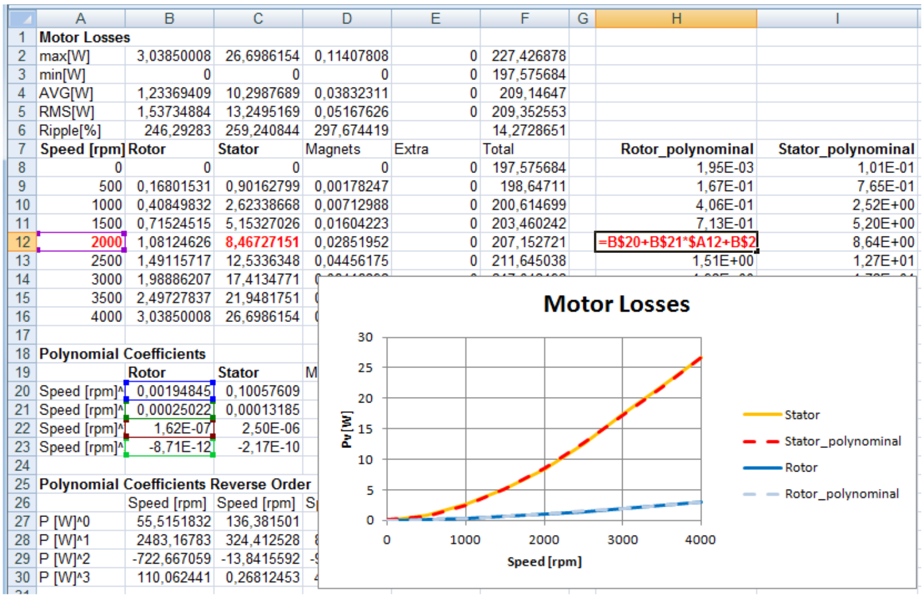


Fig. 119: Loss Calculation - Results

3.6.8.1 Polynomial Coefficients

For the loss curves, the coefficients (xy and invers yx) of polynomials 3rd to 7th order can be calculated and copied over the clipboard to create mathematical loss models in other applications.





	A	B	C	D	E	F	G	H	I
1	Motor Losses								
2	max[W]	3,03850008	26,6986154	0,11407808	0	227,426878			
3	min[W]	0	0	0	0	197,575684			
4	AVG[W]	1,23369409	10,2987689	0,03832311	0	209,14647			
5	RMS[W]	1,53734884	13,2495169	0,05167626	0	209,352553			
6	Ripple[%]	246,29283	259,240844	297,674419		14,2728651			
7	Speed [rpm]	Rotor	Stator	Magnets	Extra	Total		Rotor_polynominal	Stator_polynominal
8	0	0	0	0	0	197,575684		1,95E-03	1,01E-01
9	500	0,16801531	0,90162799	0,00178247	0	198,64711		1,67E-01	7,65E-01
10	1000	0,40849832	2,62338668	0,00712988	0	200,614699		4,06E-01	2,52E+00
11	1500	0,71524515	5,15327026	0,01604223	0	203,460242		7,13E-01	5,20E+00
12	2000	1,08124626	8,46727151	0,02851952	0	207,152721		=B$20+B$21*$A12+B$2	8,64E+00
13	2500	1,49115717	12,5336348	0,04456175	0	211,645038		1,51E+00	1,27E+01
14	3000	1,98886207	17,4134771						
15	3500	2,49727837	21,9481751						
16	4000	3,03850008	26,6986154						
17									
18	Polynomial Coefficients								
19		Rotor	Stator	M					
20	Speed [rpm]^	0,00194845	0,10057609						
21	Speed [rpm]^	0,00025022	0,00013185						
22	Speed [rpm]^	1,62E-07	2,50E-06						
23	Speed [rpm]^	-8,71E-12	-2,17E-10						
24									
25	Polynomial Coefficients Reverse Order								
26		Speed [rpm]	Speed [rpm]	S					
27	P [W]^0	55,5151832	136,381501						
28	P [W]^1	2483,16783	324,412528						
29	P [W]^2	-722,667059	-13,8415592						
30	P [W]^3	110,062441	0,26812453						

Fig. 120: Loss Calculation - Polynomial coefficients

3.6.8.2 Loss Calculation of Linear Motors

The calculation of losses are generally performed by using *"Fourier Transfomation"*, which are only valid for periodic signals. Therefore are the losses of linear motors only for periodic models with negative or positive periodicity performed. In smartFEM available non-periodic models can the userselect by a switch whether the model shall be periodic or non-periodic.

As far as modesl of linear motor are designed by using CAD software and imported as DXF file into smartFEM the user has to take this into account.

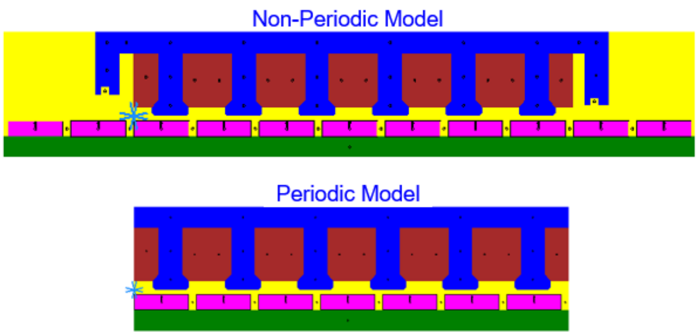


Fig. 121: Loss Calculation - Linear Motors

3.7 Ld/Lq identification

For a field-oriented control of permanent magnetically excited synchronous motors, the tables of Ld and Lq identifications, which are determined by the Park/Clarke transformation, are often used. These tables can be calculated in smartFEM using the *"Ld/Lq"* function. The first step is to create a table with fluxes and their phase shifts. This can be done manually line by line or parametrically via the dialog *"Data - Add - ..."* or parameterized. This table is generated automatically for the simulation of a field-oriented control with the program *"CASPOC"* suitable for the simulation of power electronics.

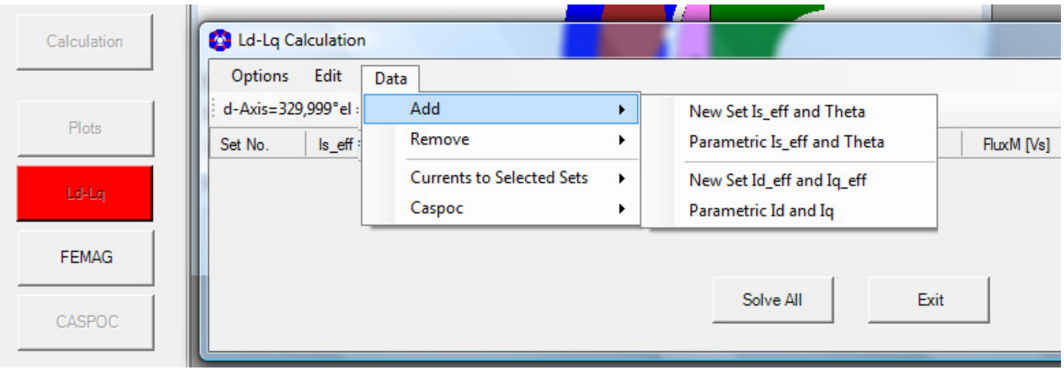
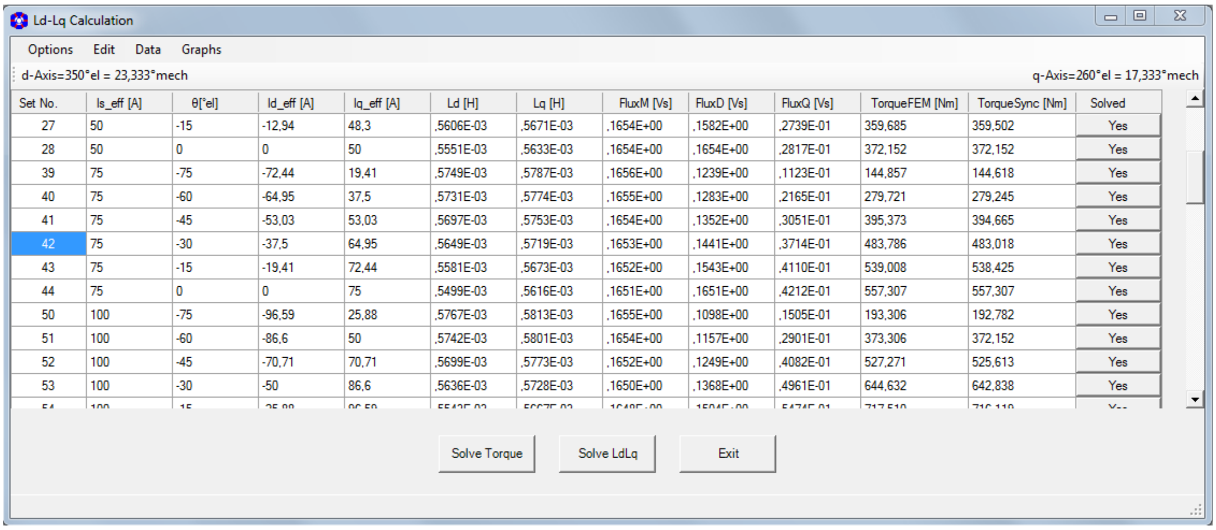


Fig. 122: Ld / Lq - Creation of data records

The records can be calculated individually or all automatically after pressing the *"Solve All"* button.



Set No.	Is_eff [A]	θ[°el]	Id_eff [A]	Iq_eff [A]	Ld [H]	Lq [H]	FluxM [Vs]	FluxD [Vs]	FluxQ [Vs]	TorqueFEM [Nm]	TorqueSync [Nm]	Solved
27	50	-15	-12,94	48,3	,5606E-03	,5671E-03	,1654E+00	,1582E+00	,2739E-01	359,685	359,502	Yes
28	50	0	0	50	,5551E-03	,5633E-03	,1654E+00	,1654E+00	,2817E-01	372,152	372,152	Yes
39	75	-75	-72,44	19,41	,5749E-03	,5787E-03	,1656E+00	,1239E+00	,1123E-01	144,857	144,618	Yes
40	75	-60	-64,95	37,5	,5731E-03	,5774E-03	,1655E+00	,1283E+00	,2165E-01	279,721	279,245	Yes
41	75	-45	-53,03	53,03	,5697E-03	,5753E-03	,1654E+00	,1352E+00	,3051E-01	395,373	394,665	Yes
42	75	-30	-37,5	64,95	,5649E-03	,5719E-03	,1653E+00	,1441E+00	,3714E-01	483,786	483,018	Yes
43	75	-15	-19,41	72,44	,5581E-03	,5673E-03	,1652E+00	,1543E+00	,4110E-01	539,008	538,425	Yes
44	75	0	0	75	,5499E-03	,5616E-03	,1651E+00	,1651E+00	,4212E-01	557,307	557,307	Yes
50	100	-75	-96,59	25,88	,5767E-03	,5813E-03	,1655E+00	,1098E+00	,1505E-01	193,306	192,782	Yes
51	100	-60	-86,6	50	,5742E-03	,5801E-03	,1654E+00	,1157E+00	,2901E-01	373,306	372,152	Yes
52	100	-45	-70,71	70,71	,5699E-03	,5773E-03	,1652E+00	,1249E+00	,4082E-01	527,271	525,613	Yes
53	100	-30	-50	86,6	,5636E-03	,5728E-03	,1650E+00	,1368E+00	,4961E-01	644,632	642,838	Yes

Fig. 123: Ld / Lq result table

Various diagrams can then be output to the result table.

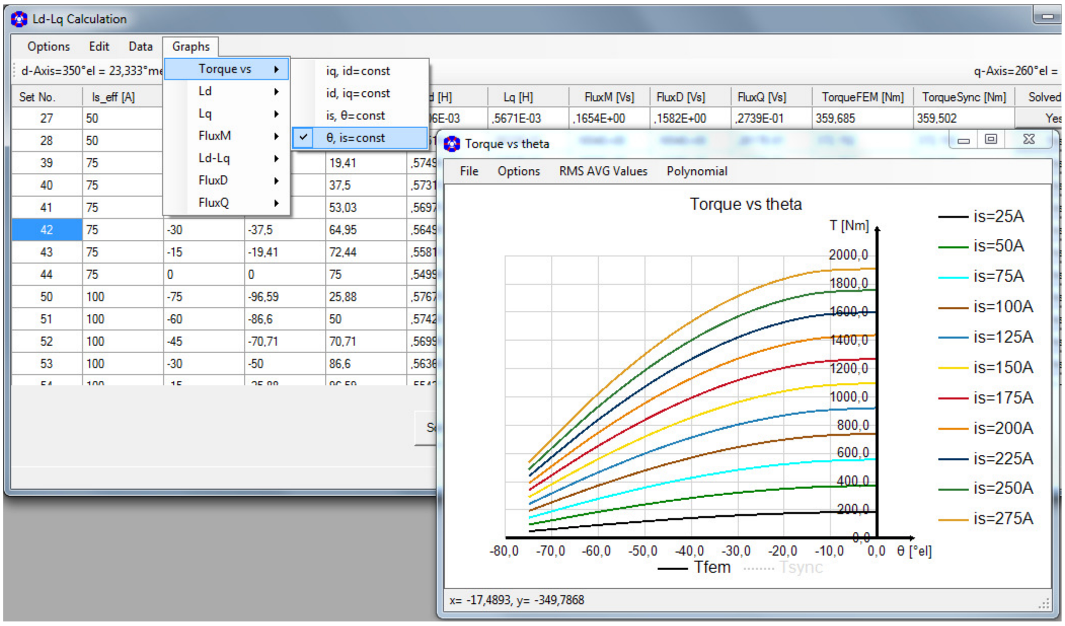


Fig. 124: Ld/Lq - Diagram Torque vs. Theta with is = constant

The phase diagram can be displayed for each selected table line.

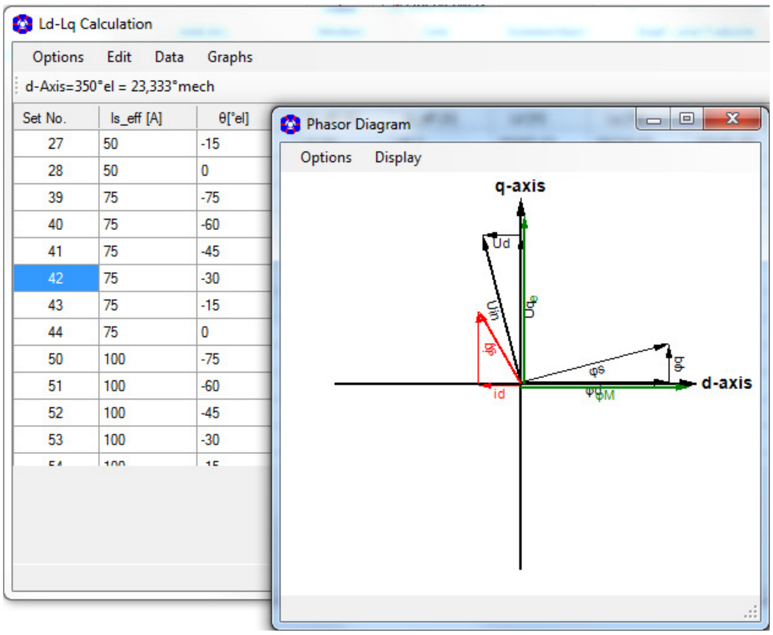


Fig. 125: Ld/Lq - Phase diagram

The results table can be transferred and used in other software programs via clipboard by *"Options - Copy Sets"*.

A result table generated for the CASPOC program is stored in the xml formatted file via the *"Data Caspoc Export File"* dialog, which is used by CASPOC as a look-up table for electronic simulation.

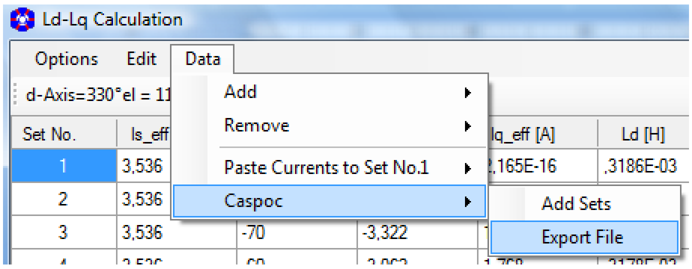


Fig. 126: Ld / Lq - Creation of a result file for CASPOC

3.8 Calculation of Special Effects

For the calculation and investigation of specific effects (for example, the resting moment caused by a single magnet), it is possible to assign different material properties to individual surface elements, like for example:

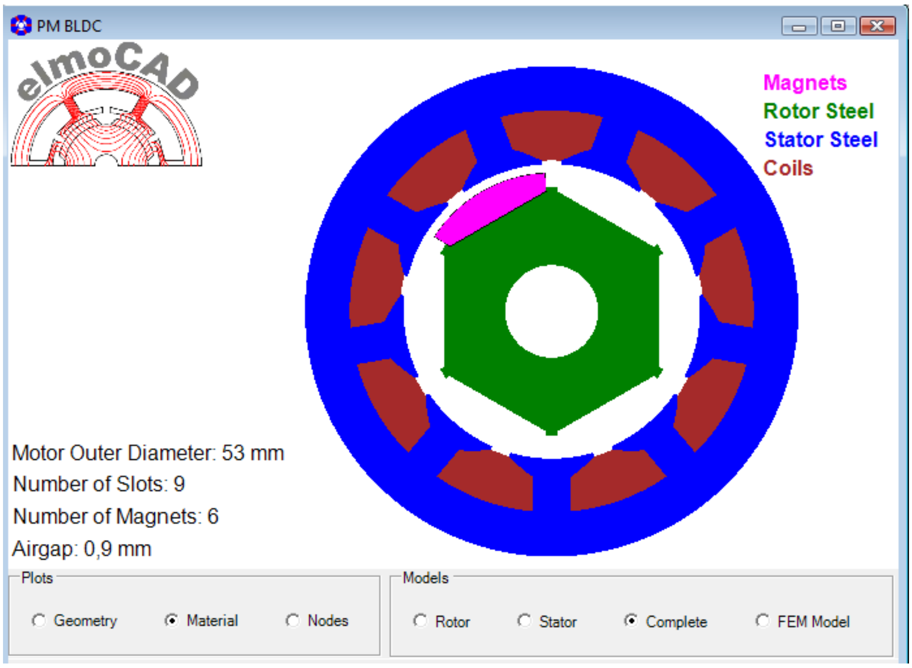
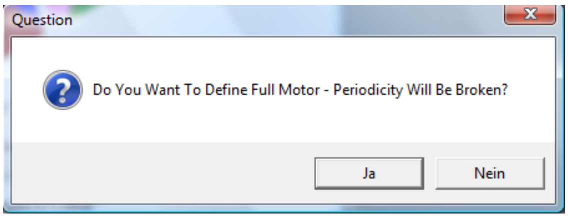


Fig. 127: Simulation model with a magnet

Note:
As a result of this, the motor model no longer satisfies the periodic symmetrical conditions specified by the number of magnets and slots in the rotor and stator topology. The results therefore only refer to this special case and cannot be transferred to the complete periodic simulation model. User is asked for acknowledgement in a pop-up-window.



The change in the material properties of individual surface elements is achieved by clicking on the corresponding surface element with the left mouse button in the representation *"Geometry"* or *"Material"* while simultaneously pressing a key combination *"CTRL", "ALT", "Shift":*

- CTRL → Rotor Iron
- ALT → Stator iron
- ALT + Shift → Magnet
- ALT + CTRL → Air

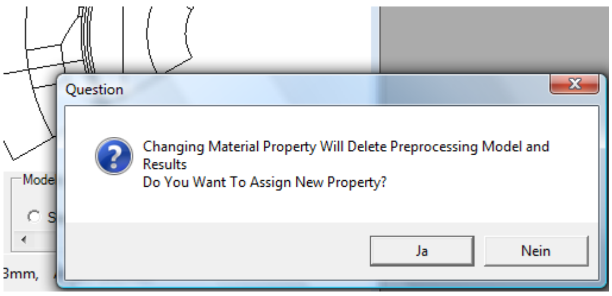


Fig. 128: Change of the material properties of individual surface elements

When using this function, it is important to know that when this function is being performed, all results will reset, and the question pops up whether an "entire" motor model is to be built up.

3.9 Graphical display and evaluation of results

The results are displayed automatically in a new window in graphics form after a calculation is started by clicking on the *"Calculation"* button and performed. If calculations have already been carried out, this is indicated by the *"Results"* button in the upper menu line. By clicking on this button, the window with the graphics is shown.

The individual graphics are selected by clicking on the corresponding line in the menu of the graphics window on the right side. When a check sign is set in a square, the corresponding graphic is automatically included in the report when the project report is generated.

The *"Options"* menu allows you to copy graphics as well as the calculated values via the buffer to other applications and to edit them there.

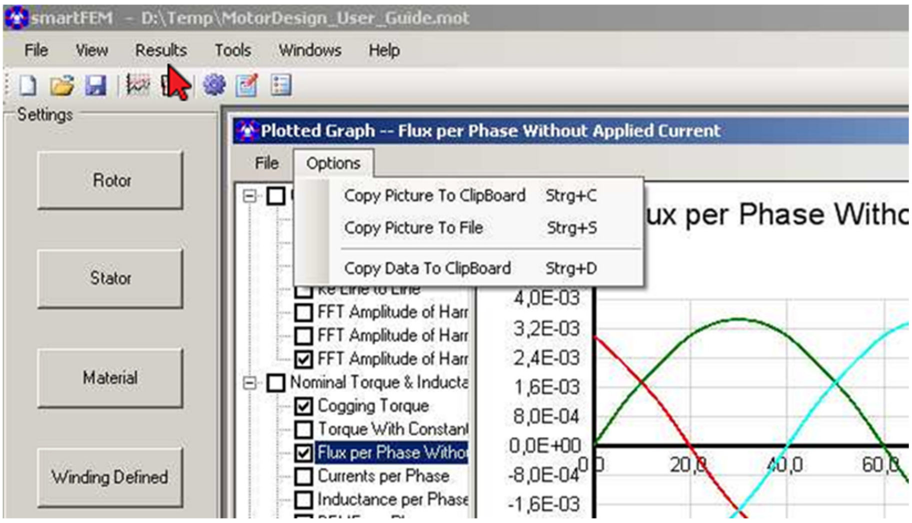


Fig. 129: Graphical display and evaluation of the results

3.9.1 Results

The calculation results are displayed as graphics after clicking the *"Results"* button. Both graphics and values can be copied to other applications via *"Options"* → *"Copy Picture*" or *"Copy Data".* The following graphics can be displayed:

3.9.1.1 Cogging Torque and BEMF

• Cogging Torque + BEMF

- Cogging moment and harmonic
- Flow per phase without/with current and harmonic for phase U
- Inductance per phase
- BEMF per phase/string and harmonic for phase U/UV
- ke per strand

• Expected values as an analytical calculation of the expected values for

- Expected motor torque
- Cogging Torque
- Phase current, phase-element current
- BEMF per phase

3.9.1.2 Nominal Torque and Inductance

The following calculation results can be displayed:

- Nominal Torque + Inductance
  The values are calculated at rotor positions within a BMEF period with the step size specified in *"Motor Parameters".*

  - Torque
  - Cogging Torque
  - Reconstructed Cogging Torque
    (transformation of the results for the cogging torque at rotor positions of torque to the rotor positions within the cogging torque period)
  - Phase current, line-to-line current
  - Flux per phase without/with current and harmonics of phase U
  - Inductance per phase
  - BEMF per phase/line-to-line and harmonics of phase U/UV
  - ke, kt per line-to-line
  - Terminal voltage per phase/ line-to-line and harmonics of phase U/UV
  - Transformer voltage per phase and harmonics of phase U

3.9.1.3 Dynamic

The following graphics can be displayed in detail:

- Dynamic Calculation
  - Cogging torque
  - Magnetic flux without current
  - Phase currents
  - Inductors
  - BEMF
  - Magnetic flux with current
  - Number of revolutions
  - Strand currents

3.10 Saving and loading simulation models

Via the menu sequence *"File -> Save"* and *"File Save As"* the simulation models can be stored in any directory. The directory specified in the smartFEM Settings is displayed as the default folder. All parameter values and results are stored in a "modellname.mot" file. Thus, each simulation model is consistent in itself. Additionally, the used topology and material files are also stored.

There are several ways to open simulation models:

- "File -> Open"
  The location of the model can be selected and the model opened. The topologies contained in the topology directory are automatically loaded. If parameter values (dimensions, node densities, etc.) should not be displayed correctly in newer versions of the topologies, the model can also be opened with the topologies stored in the model.

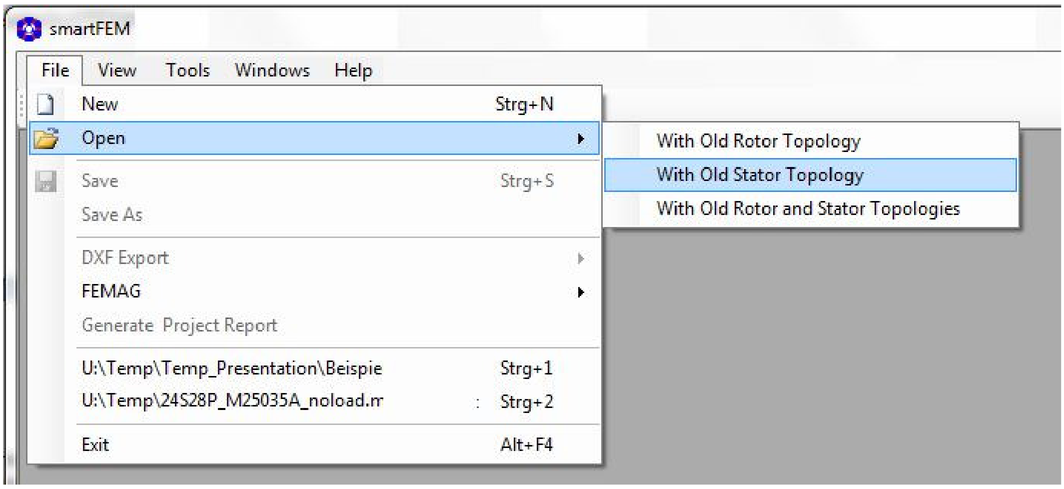


Fig. 130: Open a previous saved simulation model

- With "Ctrl + #" the corresponding simulation model is opened directly. The last four stored model files and their paths are displayed.

4 Add-ons

Add-ons are special modules that are created as the dll file type and stored in the *“... \ smartFEM \ ElTopology”* directory. They are recognized and listed dynamically by smartFEM when the *"Add-ons"* menu is selected in *"Motor Parameters"*. Users can also create their own dlls, if they know the interface.

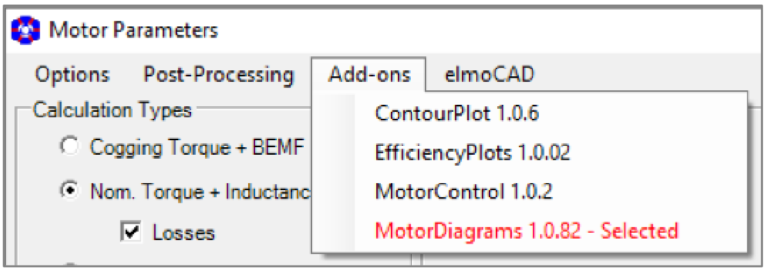


Fig. 131: Add-ons - Call-up menu

4.1 Motor Diagrams

The *"MotorDiagrams"* module is a special module, with which motor diagrams with different graphics such as *"Torque-speed-diagram"* can be generated. The basis for this is an Ld/Lq matrix, whereas the results of individual Ld/Lq sets in mathematical models with polynomials can be used for creation of diagrams.

The call up of the module motor diagrams is made in *"Motor Parameters"* via the *"Add-Ons - MotorDiagrams"* menu.

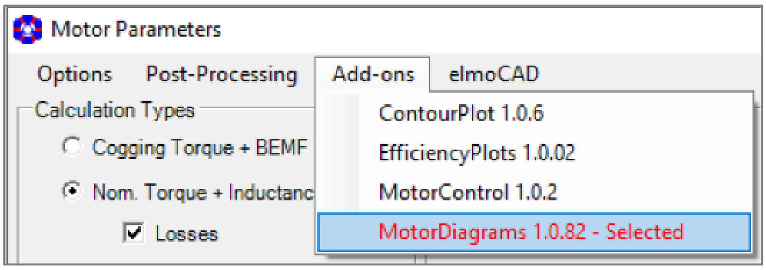


Fig. 132: Motor diagrams - Call up

The calculations for generating different graphics are performed on the basis of a current-theta matrix specified by the user.

From the current theta matrix, corresponding sets are generated for the Ld/Lq calculations. In addition to the Ld/Lq identification, different additional results are calculated for each Ld/Lq set.

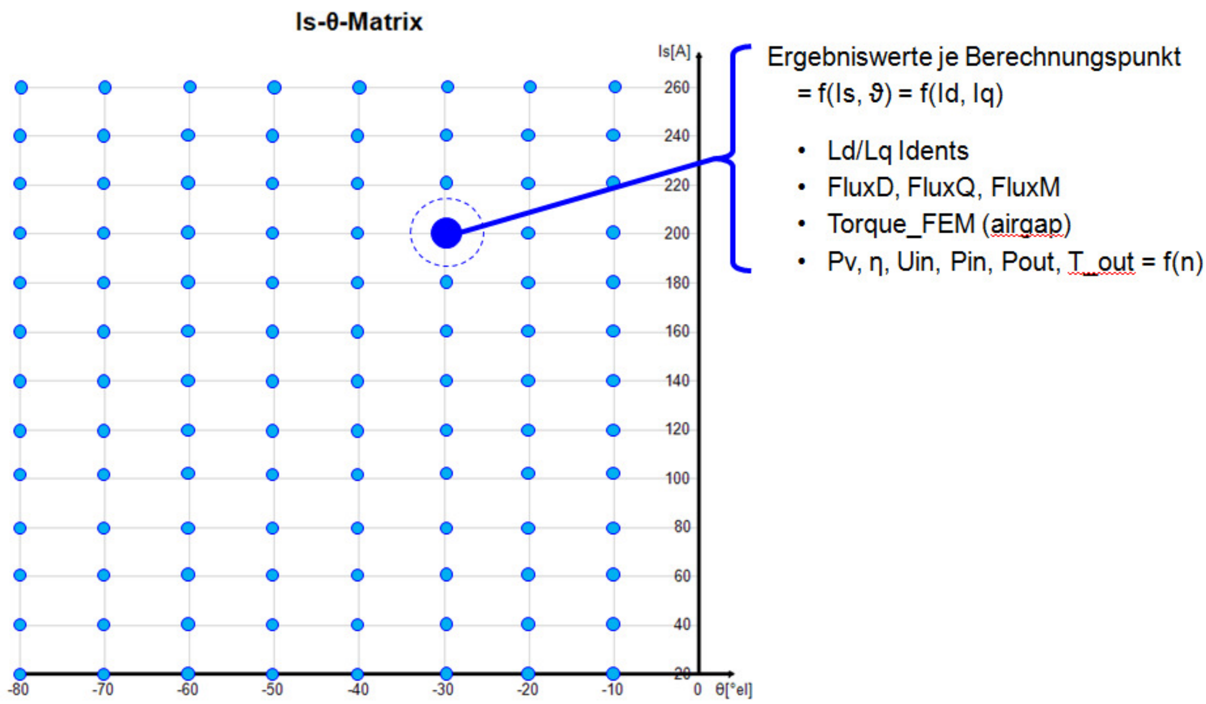


Fig. 133: Motor diagrams - Current theta matrix and result values

In the *"Diagram"* screen, the parameters for generating the graphs are computed with the calculation of the graphs to be displayed.

The winding connection "Star/Delta" is automatically taken over from the parameter specification in the *"Motor Parameters"*.

If "0" (zero) is entered, the respective values are taken from the results of the *Nominal Torque Calculation* and the *Motor Parameters* and displayed.

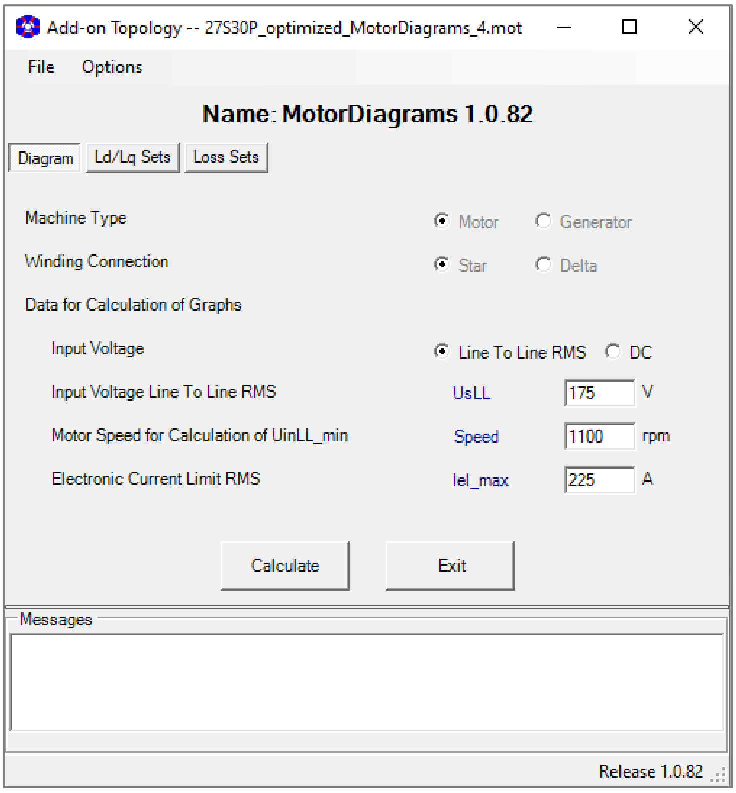
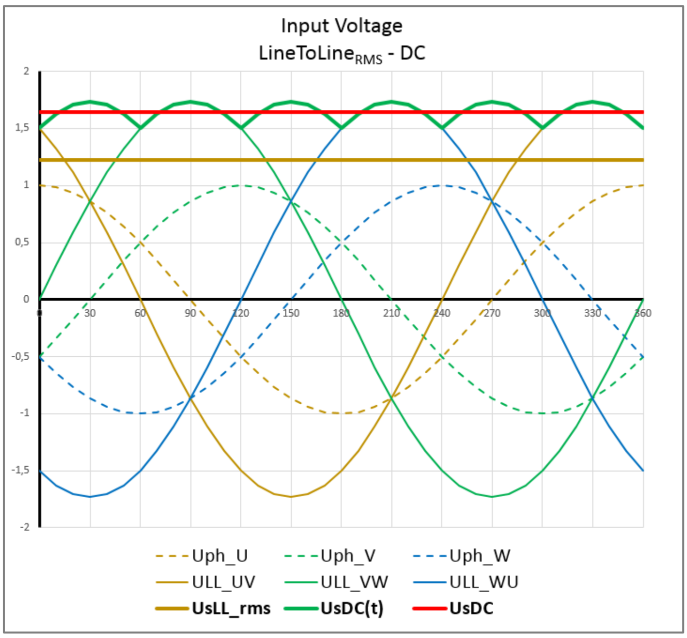


Fig. 134: Motor diagrams - Parameters for the calculation of the graphic data

When *Input Voltage = DC* is selected, the corresponding *"Input Voltage Line to Line RMS"* is determined from the input value *UsDC* and used for the calculations of the motor diagrams:



$$UsDC = Mittelwert(UsDC(t))$$

$$= \hat{U}sLL * \frac{1}{T} * \int_{\frac{\pi}{3}}^{\frac{2\pi}{3}} \sin(t)\,dt\,, \quad T = \frac{\pi}{3}$$

$$= \frac{3 * \sqrt{2}}{\pi} * UsLL_{\_rms}$$

$$\cong 1{,}35 * UsLL_{\_rms}$$

$$\Rightarrow \quad UsLL_{\;rms} = \frac{\pi}{3 * \sqrt{2}} * UsDC$$

$$\cong 0{,}74 * UsDC$$

Fig. 135: Motor diagrams - Conversion of UsDC <> UsLL_rms

The parameters for currents and phase shifts from which the Ld/Lq sets are generated for the calculation of result values of the theta current matrix are entered in the *"LdLq Sets"* screen window. The *"Simple"* type is used to calculate simple motor diagrams for individual operating points.

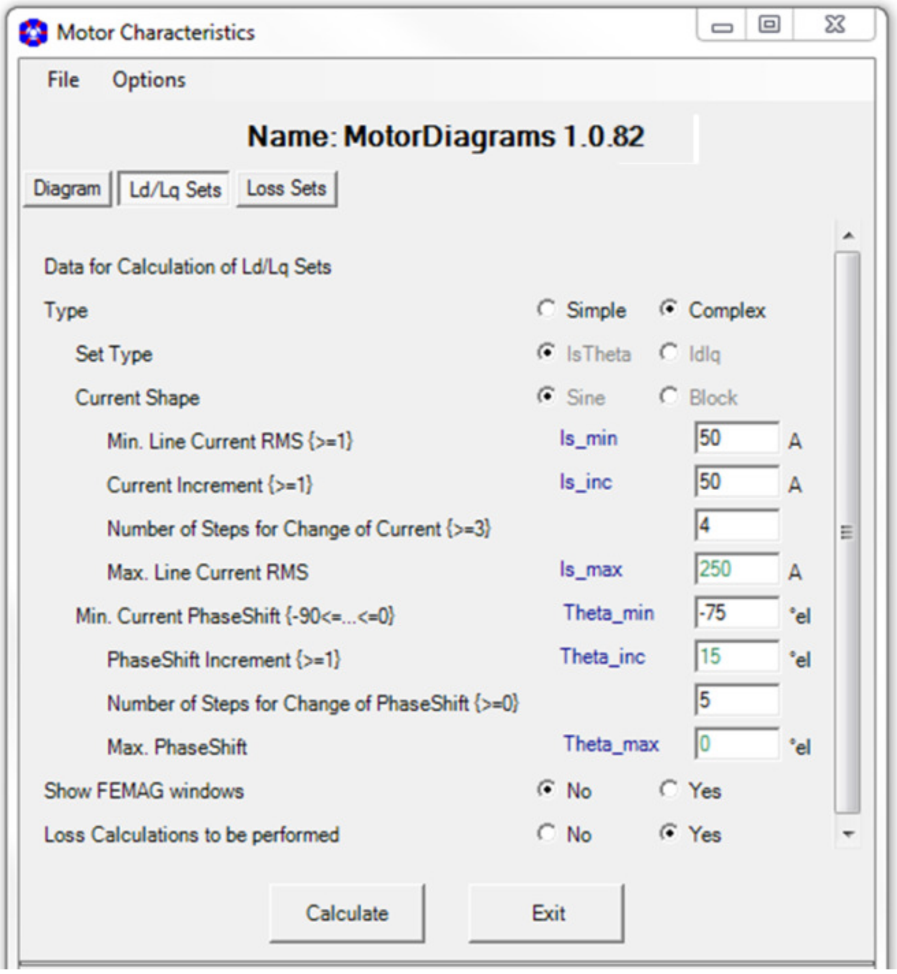


Fig. 136: Motor diagrams - Parameters for the creation of the Ld/Lq sets

For calculation of losses, the required parameters can be entered in the window *„Loss Sets“*.

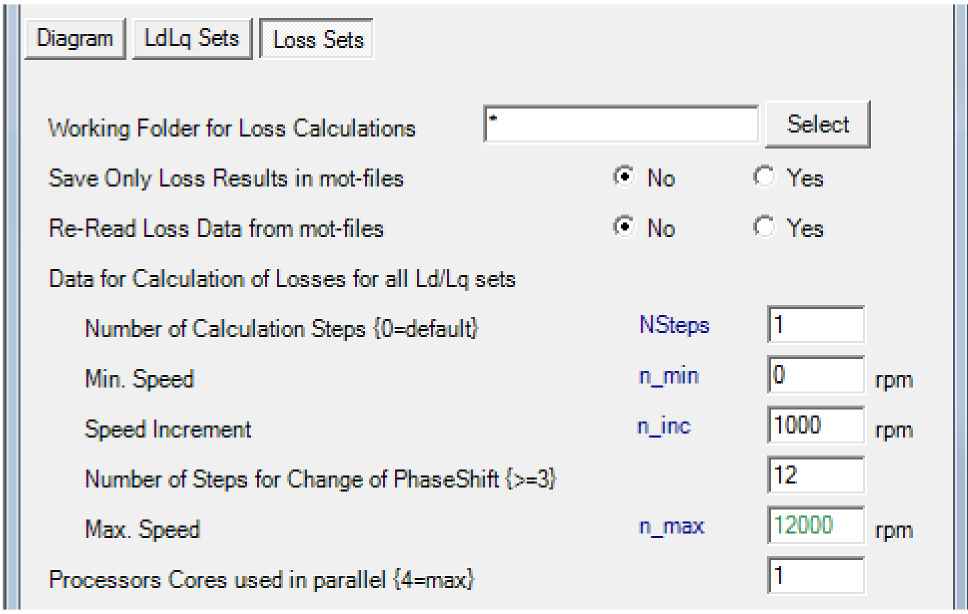


Fig. 137: Motor diagrams - Loss sets

A "Nominal Torque + Inductance Calculation" is carried out per Ld/Lq set with subsequent calculation of the losses.

After calculating the losses, these are read from the individual model files and are also stored in the "basic" model file. *„Re-Read Loss Data from Mot Files = Yes“* is used to read the loss data from the model files.

A model file (mot-file) is created with the entered data for each Ld/Lq set and stored in the directory specified in *"Working Folder"*. If the character "⋆" is indicated, the directory is used in which the "base" model opened with smartFEM is also stored. The respective filenames are supplemented by the Ld/Lq parameters Is and Theta for distinction.

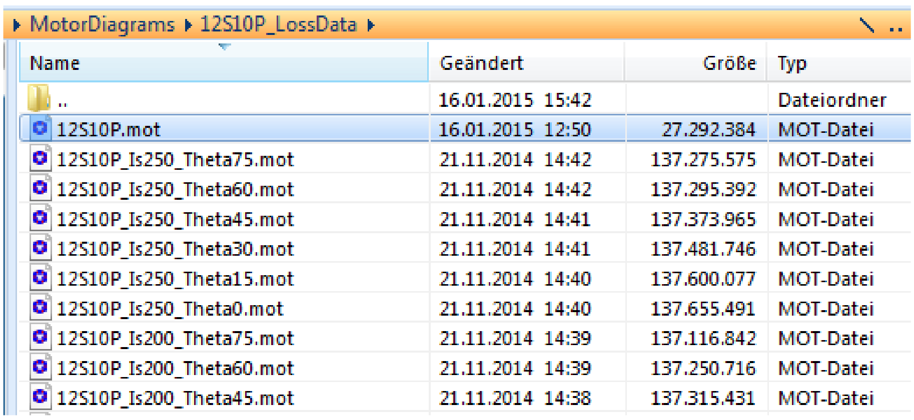


Fig. 138: Motor diagrams - Example of folder contents with "Base" and "Loss" models

4.1.1 Performing calculations

By clicking the button, the generation of the motor diagrams is triggered and carried out in the following steps:

Step 1: Sequential generation of the Ld/Lq sets from the data specified in the parameter window *„Ld/Lq Sets“.* It checks whether the set is already contained in the Ld/Lq table which can be called in the smartFEM main window via the *„Lq-Lq“* button and has already been calculated. If this is not the case, the set is inserted into the table and the calculation of this set is started.

Step 2: When all Ld/Lq sets are calculated, the loss calculations are started (i.e. generation of the model files (mot files) per Ld/Lq set and execution of the *„Nominal Torque + Inductance Calculations“* with subsequent calculations of the losses Parallel processes corresponding to the number of processor cores to be used.

In the window *„smartFEM MultiRun“* an overview of all Ld/Lq sets and their current state with regard to loss calculations is displayed. Here, the number of processor cores to be used can also be adapted from 1 to maximum of the cores present in the particular computer.

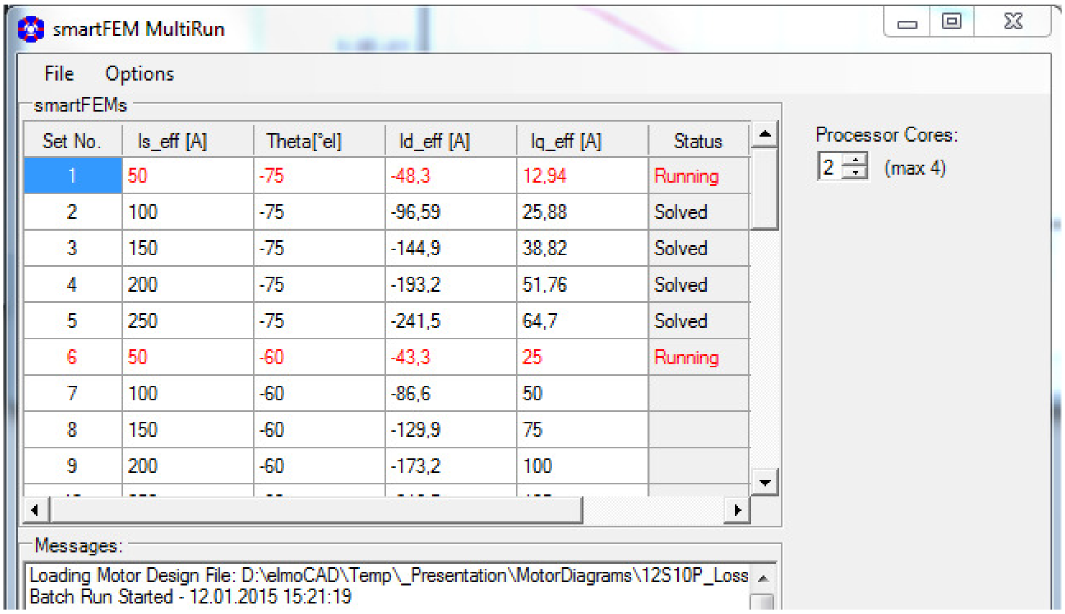


Fig. 139: Motor diagrams - smartFEM MultiRun with overview of the current processing status for the loss calculations

The smartFEM processes started with *“MultiRun”* run automatically with a lower priority than the ”smartFEM-Master- Process" in order not to block it.

Step 3: After completion of step 1, the motor diagrams are generated parallel to the calculation of the losses. They can be edited by the user and are automatically complemented after the losses have been calculated.

The values entered in the parameter window *"Diagram"* are based on:

- Input Voltage Line to Line or DC
- Motor Speed for calculating the minimum voltage Input Voltage Line to Line
- Electronic Current Limiter for current limitation. This serves only for the optical delineation of the operating range, which can be achieved by the maximum motor current.

The following diagrams are generated:

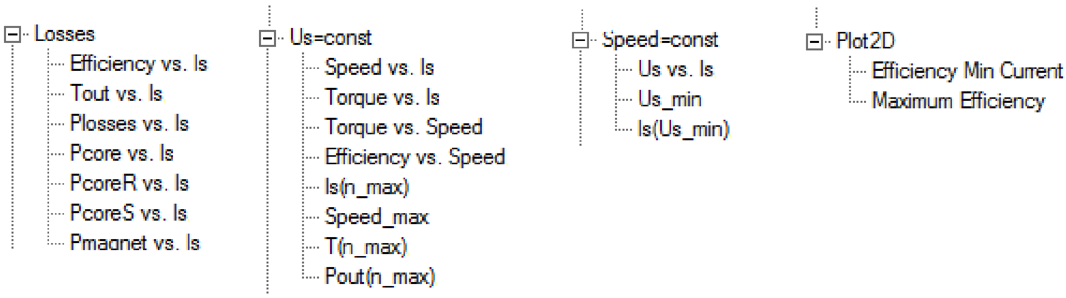


Fig. 140: Motor diagrams - Diagram types

4.1.2 Diagram examples

4.1.2.1 Torque vs. number of revolutions

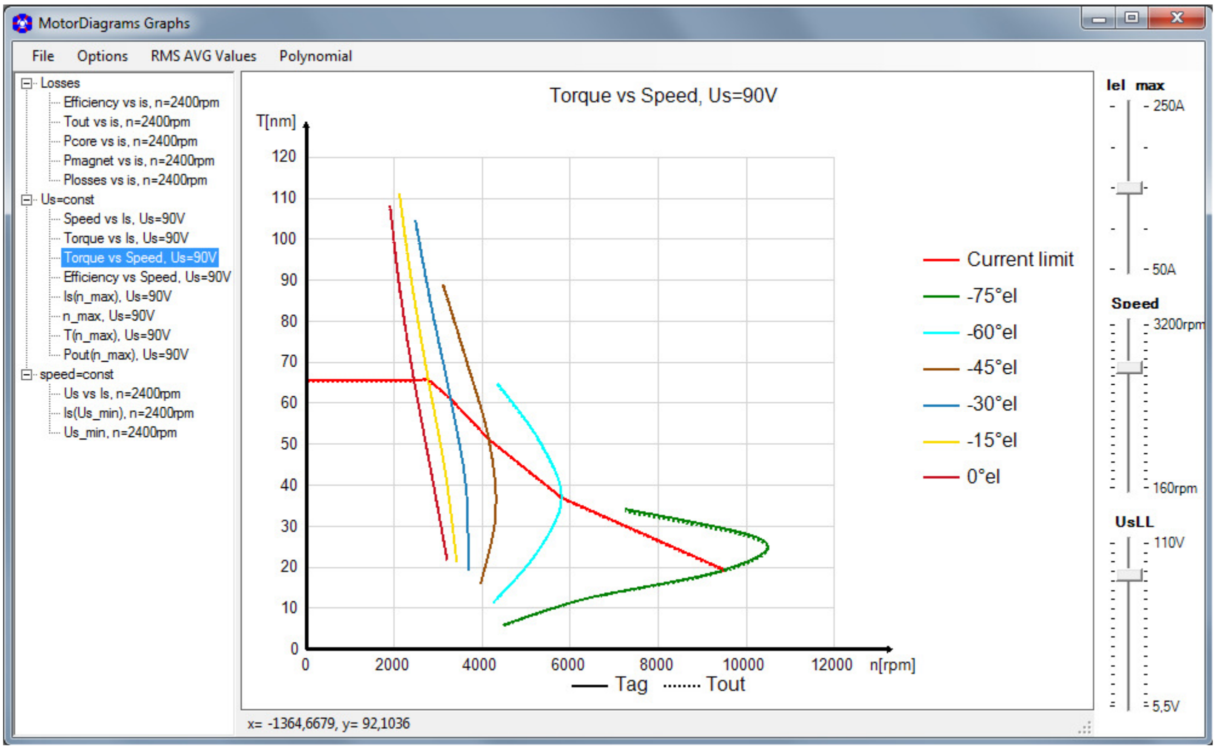


Fig. 141: Motor diagrams - Torque vs. torque number of revolutions

The data entered in the parameter window *"Diagram"* can be changed after creating the diagrams with the help of slides shown in the diagram windows, whereby the diagrams are updated immediately.

The dotted lines represent the curve progressions after deduction of the iron, magnetic and external losses.

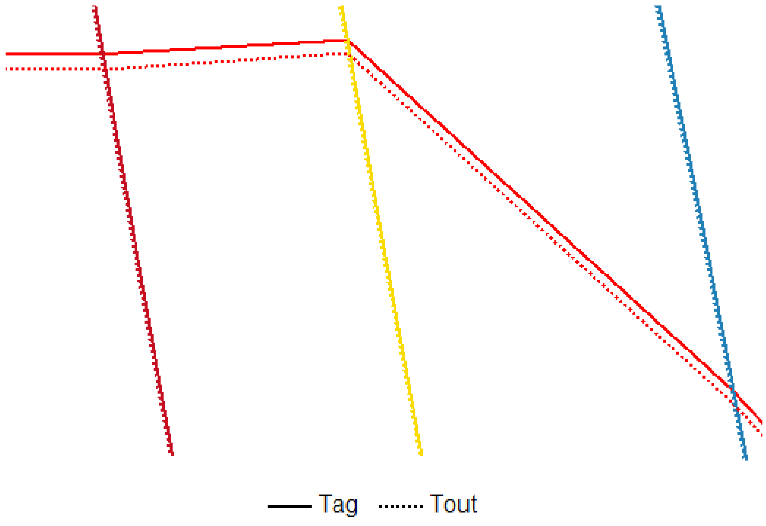


Fig. 142: Motor diagrams - Torque vs. Speed with air gap torque (Tag) and output torque (Tout)

By use of the middle mouse button or the scroll wheel individual graphs or groups can be highlighted.

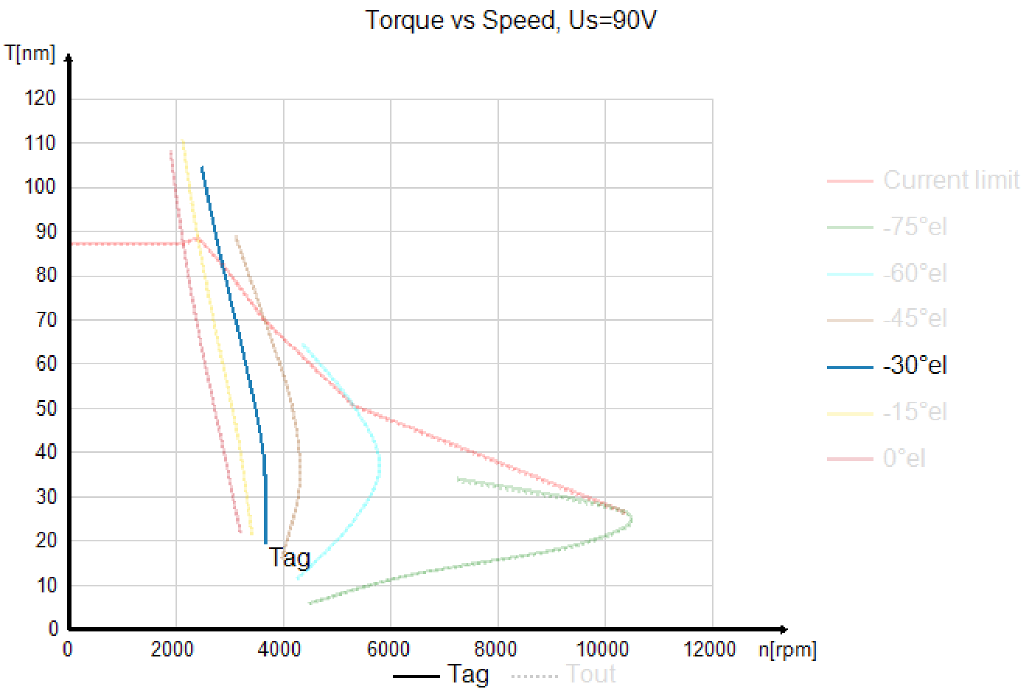


Fig. 143: Motor diagrams - Example of highlighting a single graph

Each graph can be detached from the tree diagram for user-specific placement on the screen.

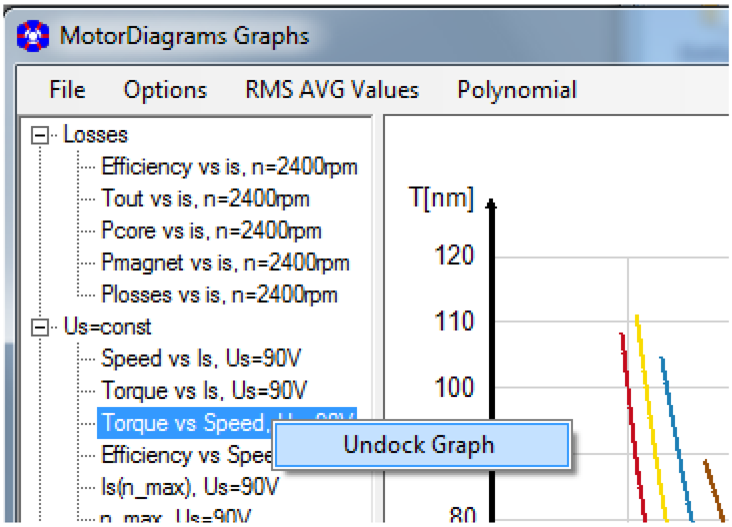


Fig. 144: Motor diagrams - Example for "Undock Graph"

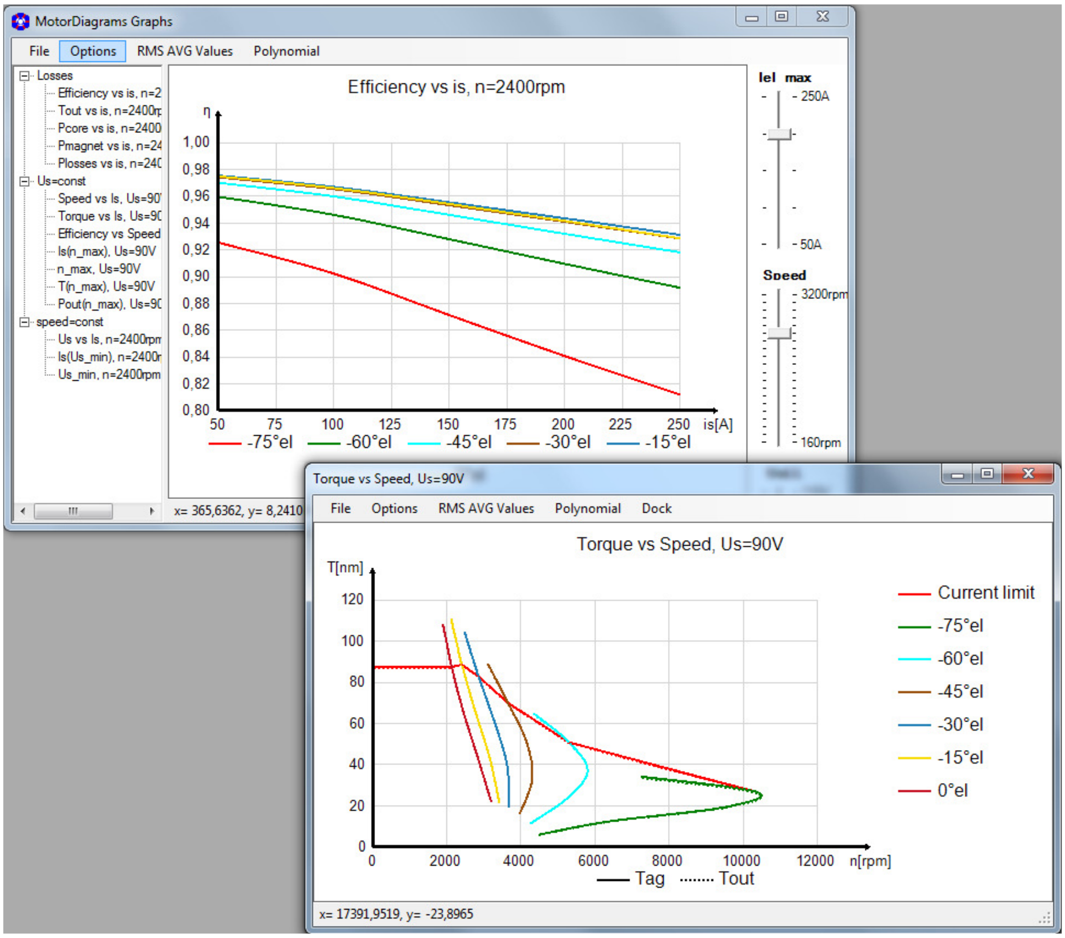


Fig. 145: Motor diagrams - Example for "Undock Graph"

In some diagrams, graphs are also available for minimum values of current and voltage.

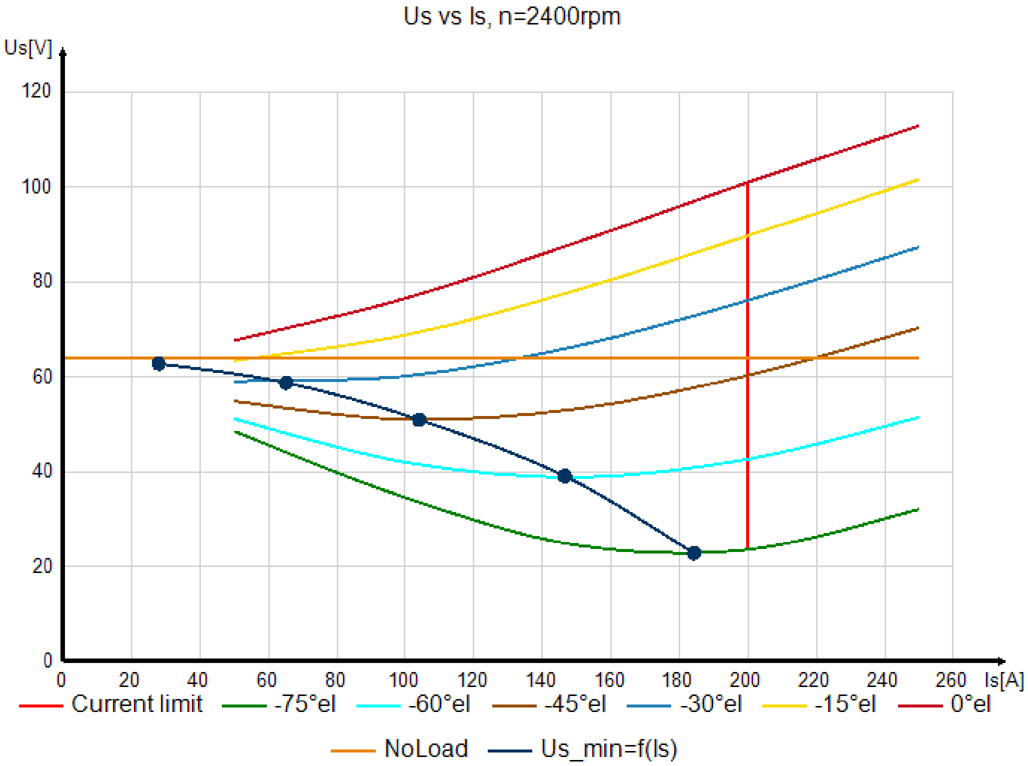


Fig. 146: Motor diagrams - Minimum values of the voltage as a function of the current

4.1.2.2 Efficiency diagrams as contour plots

Two different contour plots are generated to evaluate the efficiency as a function of torque and speed:

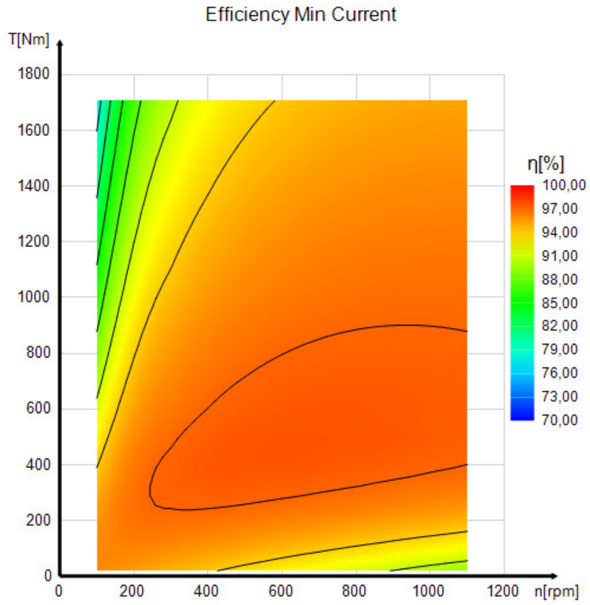


Fig. 147: Efficiency with minimum current

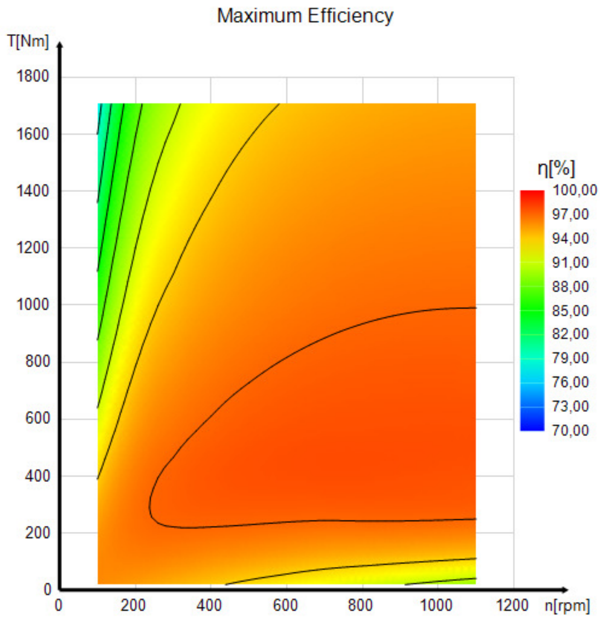


Fig. 148: Maximum efficiency

The diagrams of the respective mouse position are shown in the status line, e.g. X = 336rpm, y = 331Nm, η = 97.3%.

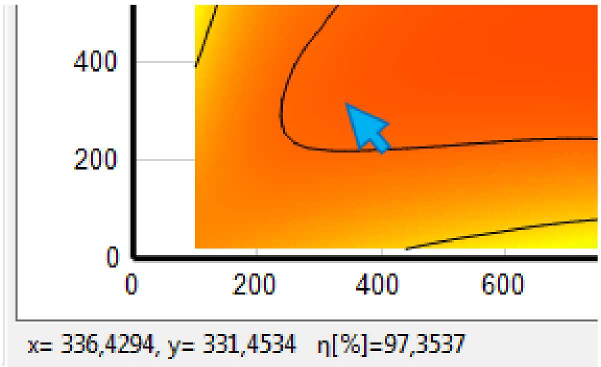


Fig. 149: Motor diagrams - Diagram values at the mouse position

By clicking the right mouse button on the plot, different settings can be made for the plot. The name *"Efficiency"* cannot be changed and it is used only for smartFEM internally.

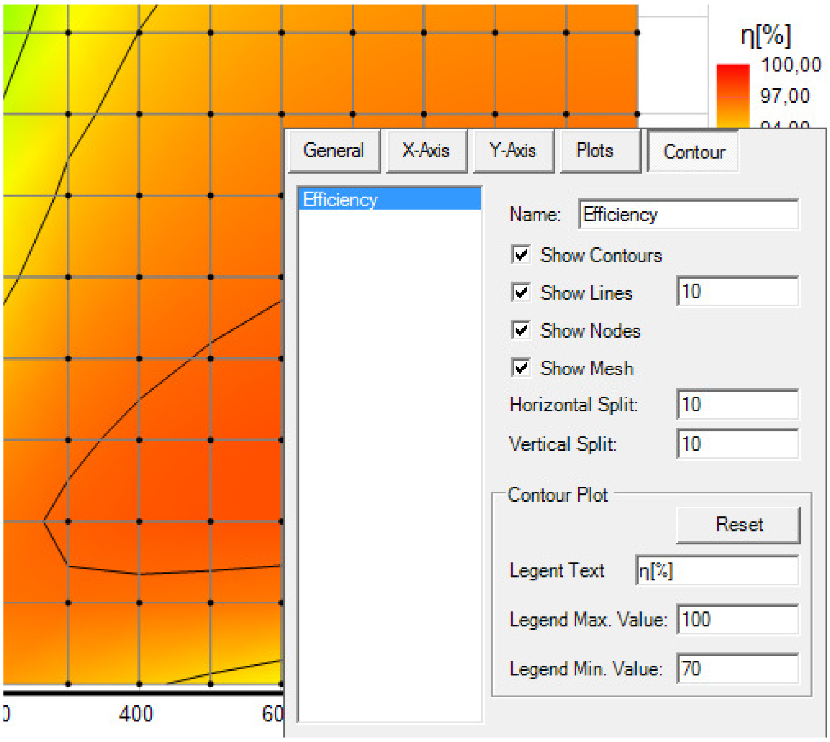


Fig. 150: Motor diagrams - Settings for contour plots

4.1.2.3 Torque Speed Feed Loop Table (MTPA - **M**aximum **T**orque **p**er **A**mpere)

For the determination of the feed setting parameter Us, Id/Iq, Ld/Lq and Theta of the motor in dependence of torque and speed with minimum current are the results via the menu Menu *„Options - Copy Torque Speed Feed Loop Table“* calculated.

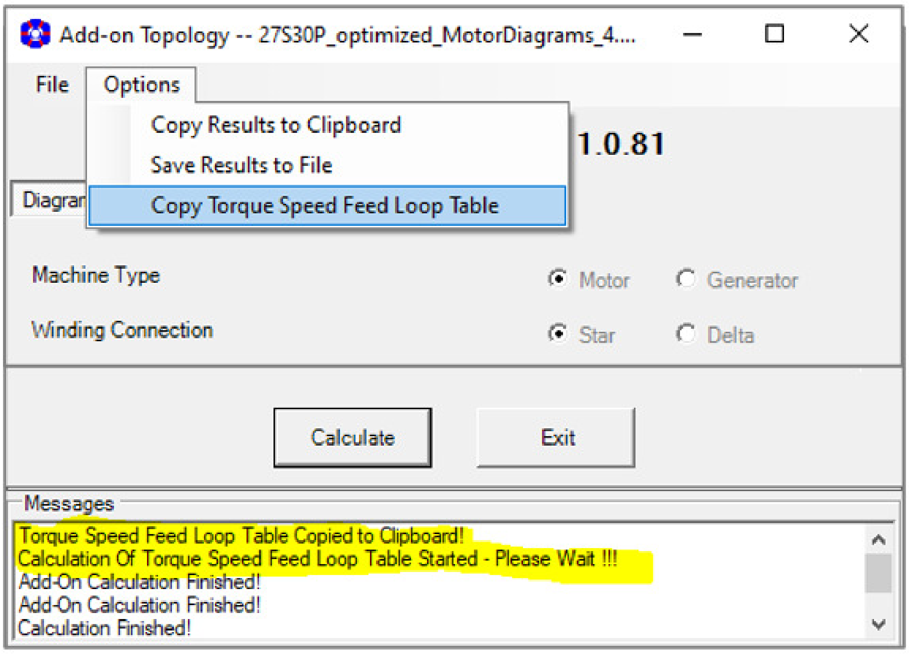


Fig. 151: MotorDiagrams - Copy Torque Speed Feed Loop Table

When the ”*Torque Speed Feed Loop Table“* is calculated then are the result data available in the clibboard and can be copied into other applications.

	A	B	C	D	E	F	G	H	I	J	O	P
1	Index	Tout [Nm]	n[rpm]	UsLL [V]	η [%]	Is_eff [A]	Id_eff [A]	Iq_eff [A]	Ld [mH]	Lq [mH]		Theta[°el]
2	1	500	100	49,4	92,7	67,5	0,0	67,5	0,552	0,562	...	0
3	2	500	200	95,5	95,9	67,6	0,0	67,6	0,552	0,562	...	0
4	3	500	300	141,7	96,9	67,7	0,0	67,7	0,552	0,562	...	0
5	4	500	400	174,9	97,3	71,1	-22,3	67,5	0,560	0,568	...	-18,28
6	5	500	500	175,0	96,3	103,3	-79,6	65,9	0,572	0,579	...	-50,39
7	6	500	600	175,0	94,5	137,9	-120,3	67,5	0,575	0,583	...	-60,70
8	7	500	700	174,9	92,6	168,3	-153,2	69,6	0,577	0,585	...	-65,57
9	8	500	800	174,9	91,3	192,6	-179,2	70,6	0,578	0,586	...	-68,50
10	9	500	900	174,9	90,3	212,3	-200,1	70,9	0,578	0,587	...	-70,47
11	10	500	1000	174,9	89,5	228,6	-217,3	70,9	0,579	0,588	...	-71,92
12	11	500	1100	174,9	88,8	242,4	-231,8	70,8	0,579	0,588	...	-73,02
13	12	1000	100	55,2	86,5	136,7	0,0	136,7	0,527	0,553	...	0
14	13	1000	200	104,2	92,4	136,9	0,0	136,9	0,527	0,553	...	0
15	14	1000	300	153,2	94,5	137,2	0,0	137,2	0,527	0,553	...	0
16	15	1000	400	174,9	95,2	146,8	-54,8	136,2	0,552	0,567	...	-21,91
17	16	1000	500	174,9	94,1	183,5	-123,5	135,7	0,565	0,576	...	-42,30
18	17	1000	600	175,0	92,4	227,7	-181,8	137,1	0,570	0,580	...	-52,97
19	18	1000	700	175,0	90,7	270,8	-233,8	136,6	0,573	0,582	...	-59,71
20	19	1500	100	61,9	80,2	210,8	0,0	210,8	0,503	0,535	...	0
21	20	1500	200	114,9	88,6	211,2	0,0	211,2	0,503	0,535	...	0
22	21	1500	300	168,1	91,8	211,6	0,0	211,6	0,503	0,535	...	0
23	22	1500	400	174,9	92,1	239,6	-118,8	208,1	0,543	0,561	...	-29,71

Fig. 152: MotorDiagrams - Torque Speed Feed Loop Table

Additional to that is then in *„MotorDiagrams Graphs“* the Plot2D *„Efficiency Torque Speed Loop Table“* available.

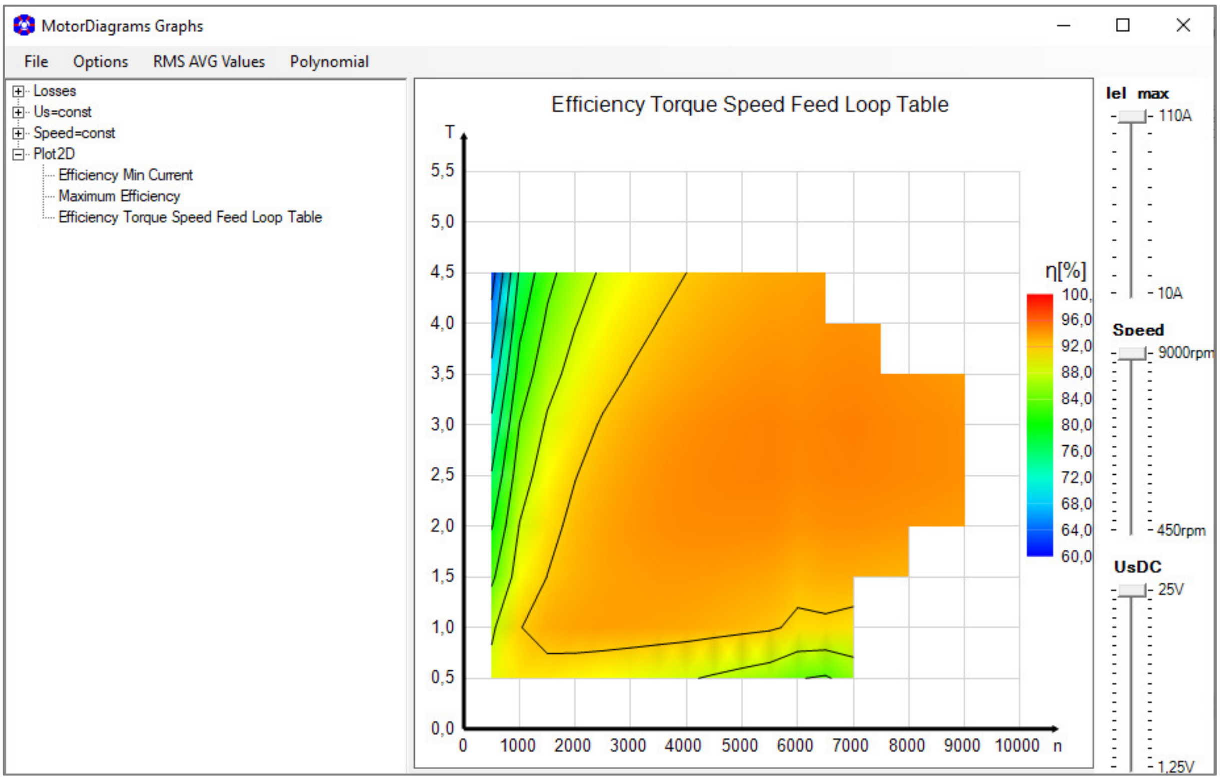
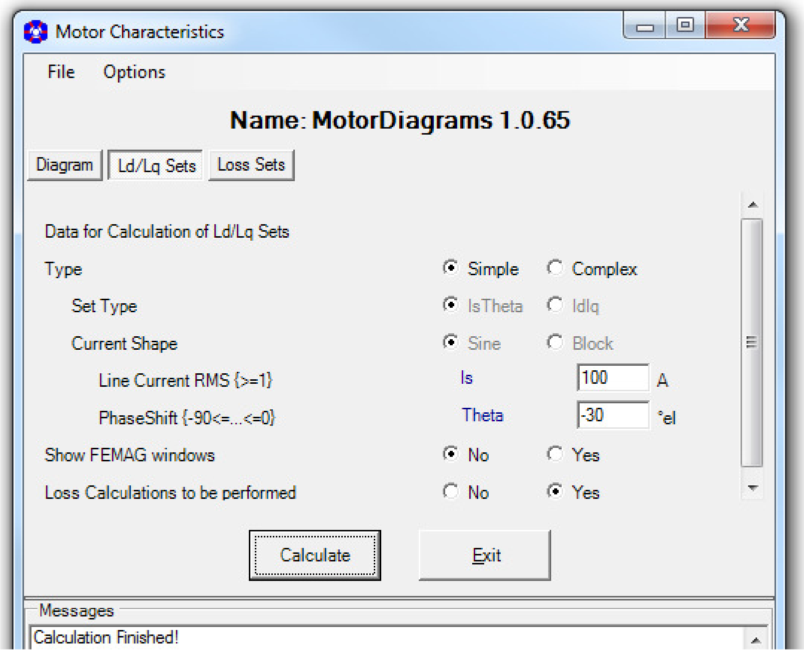


Fig. 153: MotorDiagrams - Plot2D "Effiency Torque Speed Feed Loop Table"

4.1.3 "Simple" motor diagrams

For a quick assessment of motor properties in individual operating points, "simple" motor diagrams can be generated. For this purpose *"Simple"* must be selected in the parameter group "Ld/Lq Sets", in order to enter the data for the corresponding operating point.



The following diagrams can be generated:

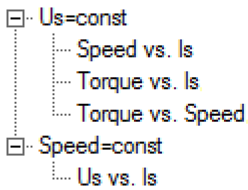


Fig. 154: Motor diagrams - Parameter input for "Simple motor diagrams"

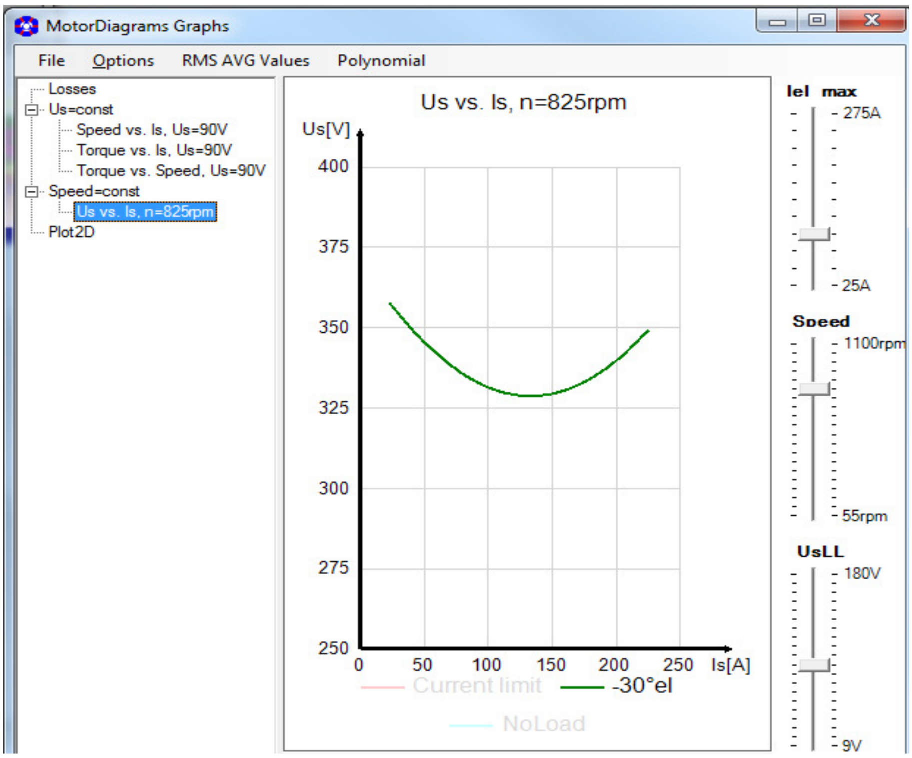


Fig. 155: Motor diagrams - Example "Simple" motor diagram Us vs. Is with Speed = constant

4.2 Motor control

The *"MotorControl"* add-on is used to generate data for the motor control electronics. After entering the calculation parameters, the unsolved Ld/Lq sets are checked and the corresponding calculations are then performed.

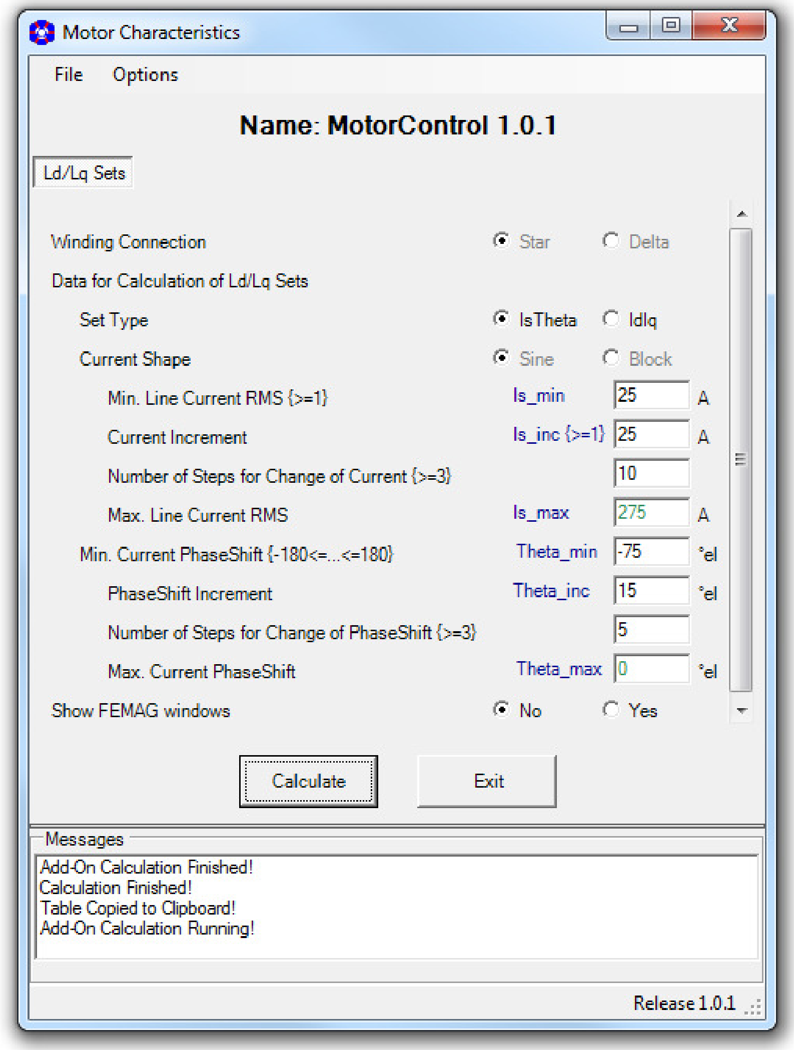


Fig. 156: Motor Control - Entering the calculation parameters

Different plots are generated, and the results can be stored both in the clip board and in a file.

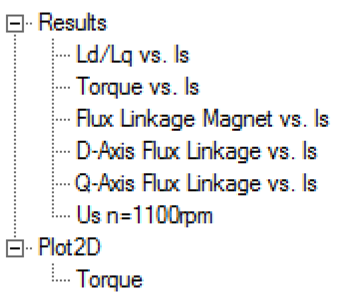


Fig. 157: Motor Control: Diagrams types

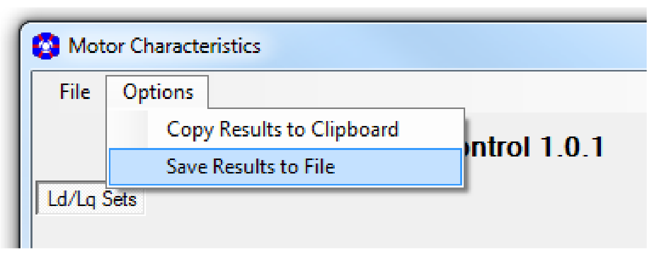


Fig. 158: Motor Control - Result memory

Diagram examples:

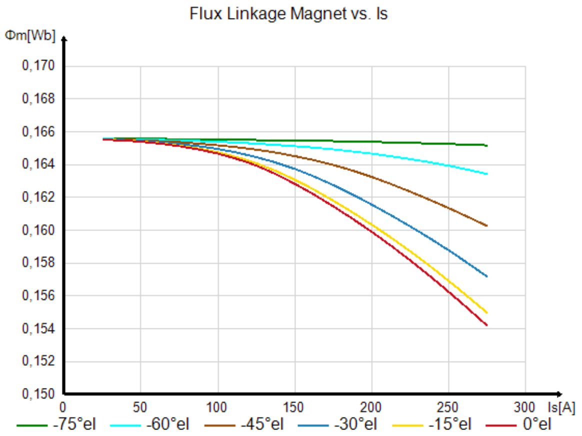


Fig. 159: Motor Control Diagram "Flux Linkage Magnet vs. Is"

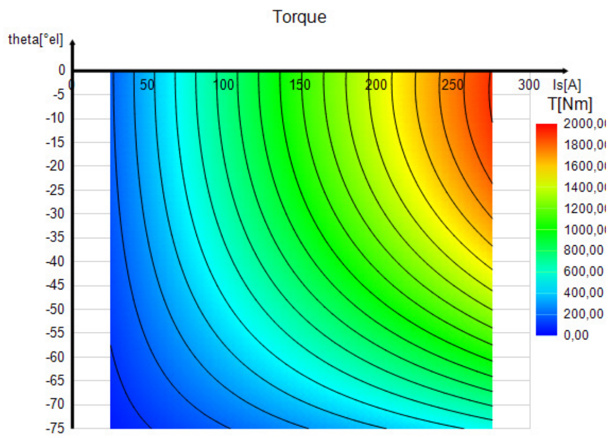


Fig. 160: Motor Control - Contour Plot "Torque vs. Theta vs. IS"

	A	B	C	D	E	F	G	H	I	J	K
1	Index	Is_eff [A]	Theta[°el]	Id_eff [A]	Iq_eff [A]	Ld [H]	Lq [H]	FluxM_eff [Vs]	FluxD_eff [Vs]	FluxQ_eff [Vs]	TorqueFEM [Nm]
2	1	25	-75	-24,1	6,5	5,665E-04	5,709E-04	1,6560E-01	1,5192E-01	3,6938E-03	48,2
3	2	50	-75	-48,3	12,9	5,719E-04	5,752E-04	1,6557E-01	1,3795E-01	7,4438E-03	96,5
4	3	75	-75	-72,4	19,4	5,749E-04	5,787E-04	1,6556E-01	1,2391E-01	1,1233E-02	144,9
5	4	100	-75	-96,6	25,9	5,767E-04	5,813E-04	1,6552E-01	1,0982E-01	1,5046E-02	193,3
6	5	125	-75	-120,7	32,4	5,777E-04	5,837E-04	1,6550E-01	9,5743E-02	1,8886E-02	242,0
7	6	150	-75	-144,9	38,8	5,786E-04	5,857E-04	1,6547E-01	8,1644E-02	2,2738E-02	290,9
8	7	175	-75	-169,0	45,3	5,791E-04	5,872E-04	1,6543E-01	6,7537E-02	2,6595E-02	340,0
9	8	200	-75	-193,2	51,8	5,796E-04	5,881E-04	1,6539E-01	5,3424E-02	3,0443E-02	389,1

Fig. 161: Motor control- Results data

4.3 General Contour Plots

The Add-On *"ContourPlot"* is used to create contour plots from any data compiled by the user.

The data of a contour plot must be stored in a tabular text file (* .txt with tab delimited columns or csv format). The first cell contains plot, column, and line names each separated by "\".

	A	B	C	D	E	F	G	H	I	J
1	i_eff\speed\eta	1000	1100	1200	1300	1400	1500	1600	1700	1800
2	10	88,22	88,46	88,61	88,69	88,72	88,7	88,65	88,58	88,47
3	11	88,62	88,92	89,12	89,25	89,33	89,37	89,37	89,34	89,28
4	12	88,88	89,22	89,47	89,65	89,78	89,86	89,9	89,91	89,9
5	13	89,02	89,41	89,71	89,93	90,1	90,22	90,29	90,34	90,36
6	14	89,09	89,53	89,86	90,12	90,32	90,48	90,59	90,66	90,71
7	15	89,1	89,57	89,95	90,24	90,48	90,66	90,8	90,9	90,98
8	16	89,06	89,57	89,98	90,31	90,57	90,78	90,95	91,08	91,18
9	17	88,98	89,53	89,97	90,33	90,62	90,86	91,05	91,2	91,33
10	18	88,88	89,46	89,93	90,32	90,64	90,9	91,11	91,29	91,43
11	19	88,75	89,36	89,86	90,28	90,62	90,9	91,14	91,34	91,5
12	20	88,6	89,25	89,78	90,21	90,58	90,88	91,14	91,36	91,54

Fig. 162: Contour Plot - Tabular text file

A single text file or a directory can be selected to generate the plots. When a directory is specified, plots of all the files contained in the directory are generated.

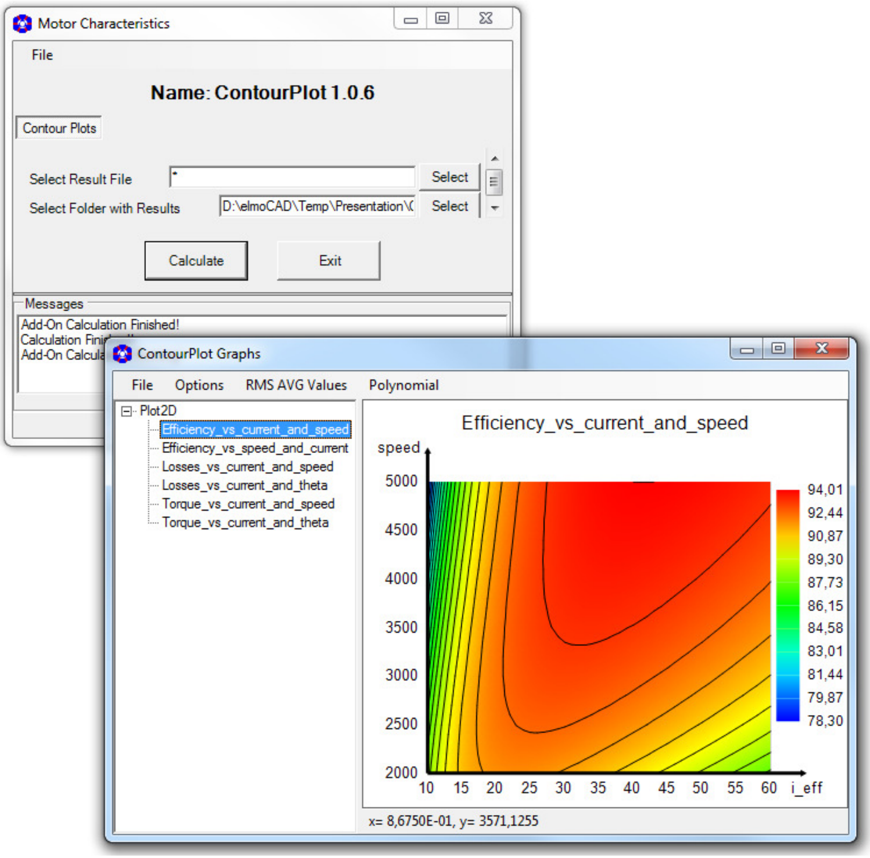


Fig. 163: Contour Plot example "Efficiency vs. Current vs. Speed"

4.4 AC-Losses

Determination of the **A**lternating **C**urrent **Losses** in stator winding which are caused by *"Skin"* and *"Proximity"* effects mainly in big electrical machines. The calculations are based on the analytical algorithms which are documented in *"Skin Effect in Large Polyphase Machines with Concentrated Windings" by Falk Laube, Helmut Mosebach und Wolf-Rüdiger Canders.*

The geometry of the stator slots and wires is assumed as rectangular. The type of winding can be single layer or double layer side by side respective on top of each other:

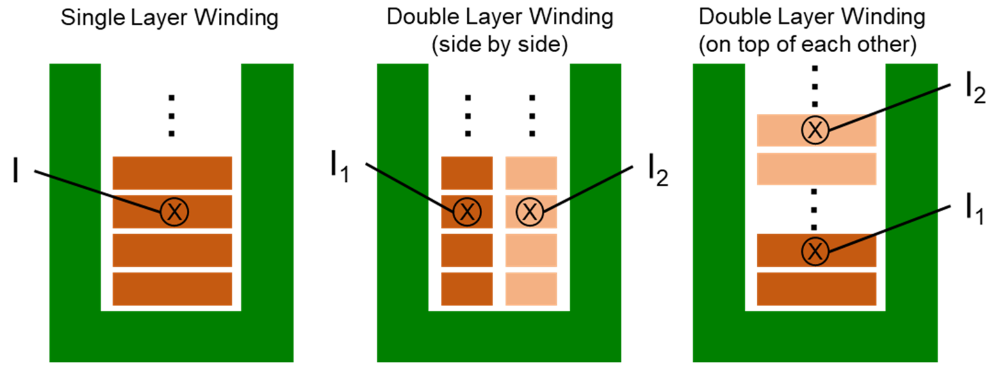


Fig. 164: AC-Losses - Geometry of slot and widings

The module is started in *"Motor Parameters"* by menu sequence *„Add-ons - AC_Losses"*. In the Tab *"Basic"* selected whether the parameter which are necessary for the calculations are entered manually or taken over from the actual opened smartFEM machine model.

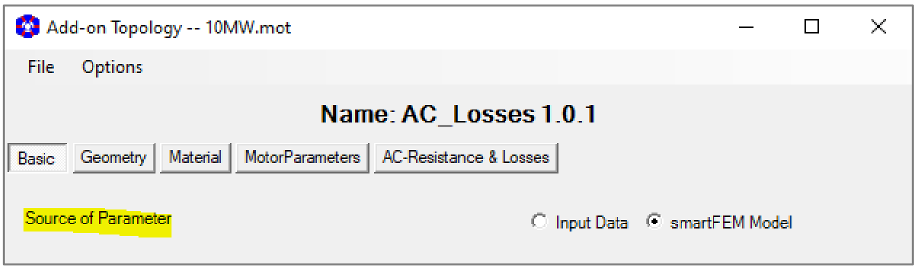


Fig. 165: AC-Losses - Source of Parameter

Hereinafter are parameter of a big machine model used which possesses nearly rectangular stator slots.

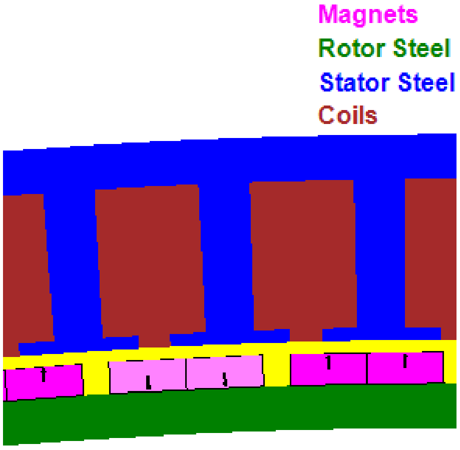


Fig. 166: AC-Losses - smartFEM model

The data which are taken from the smartFEM model are displayed in green colour and cannot be changed. Some necessary parameters are not defined in the smartFEM model and the related values have to be determined or estimated by the user and entered manually.

In Tab *"Geometry"* are these the parameter:

- Width of slots
- Number of conductors on top of each other
- Number of conductors side by side

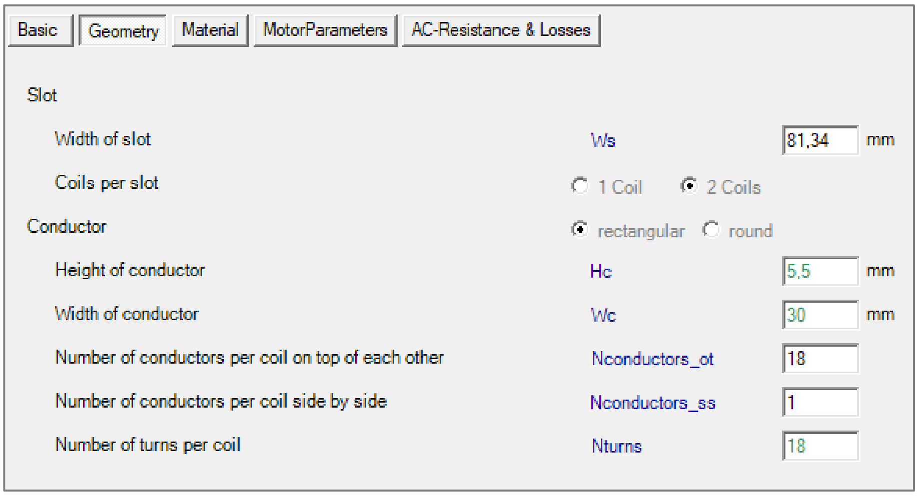


Fig. 167: AC-Losses - Geometry Parameter "smartFEM Model"

In case that in Tab *"Basic"* as *„Source of Parameter" „Input Data"* is selected all parameter except *„Number of Turns"* have to be determined and entered manually.

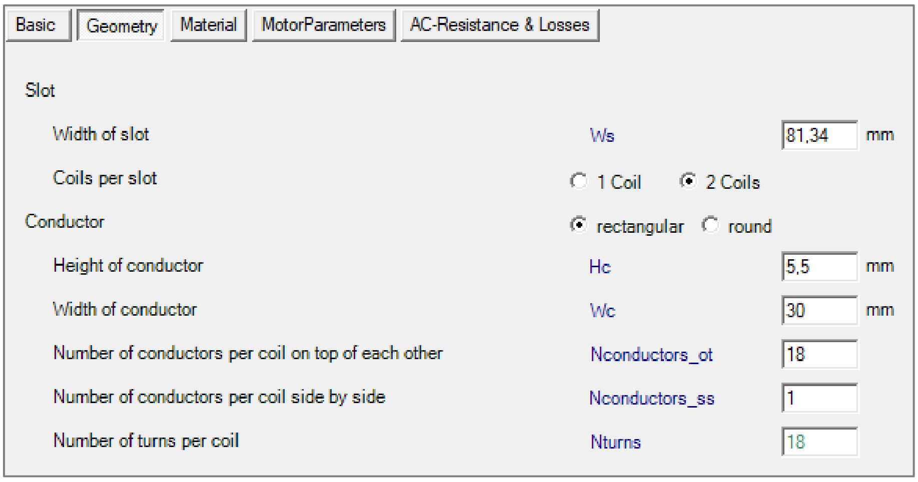


Fig. 168: AC-Losses - Geometry Parameter "Input Data"

"Material" Parameter: "Conductivity at winding temperature" is taken from *"Material"* parameter of the smartFEM model. By specification of *"Conductivity at 20°C"* the temperature of the winding is re-calculated with which the smartFEM model was calculated.

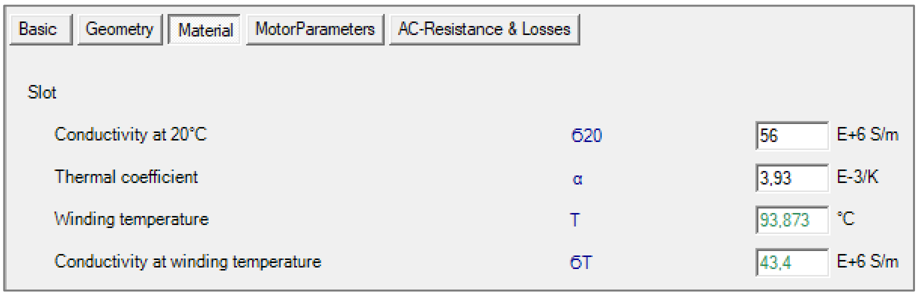


Fig. 169: AC-Losses - Material Parameter "smartFEM Model"

In case that in Tab *„Basic“* as *„Source of Parameter” „Input Data“* is selected, can the temperature of the winding entered at which the AC-Losses shall be calculated. The conductivity for this temperature is then re-calculated.

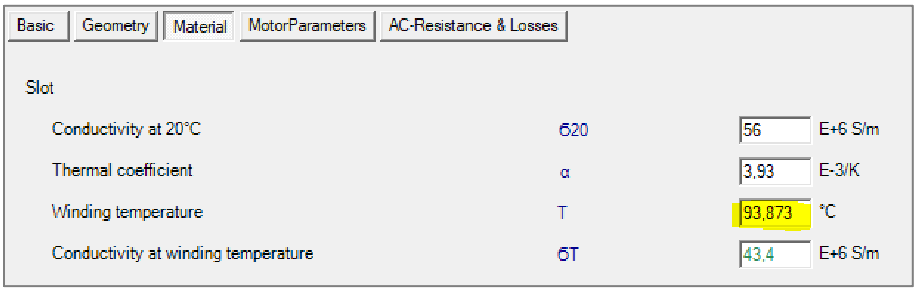


Fig. 170: AC-Losses - Material Parameter "Input Data"

„MotorParameters“ are always taken from the smartFEM model and provided as information for the user.

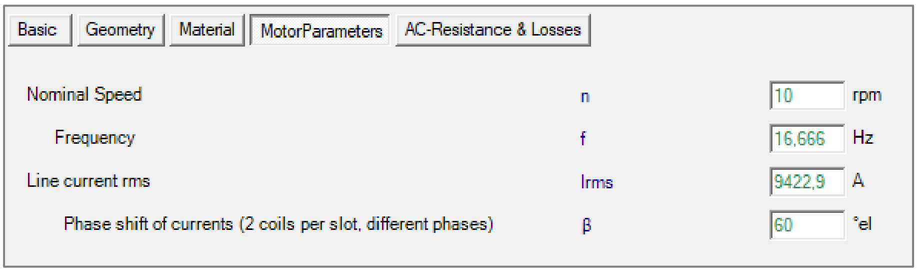


Fig. 171: AC-Losses - Motor Parameter

In the tab *„AC-Resistance & Losses“* are the results of the AC-Resistances of the windings and the AC-Losses displayed.

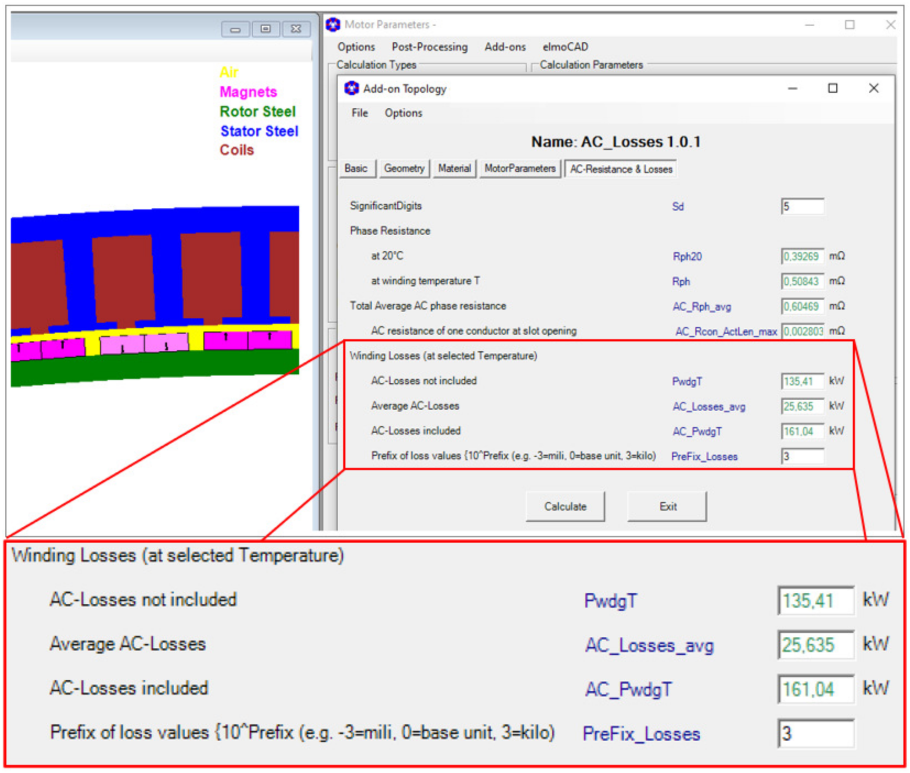


Fig. 172: AC-Losses - Results

Remark:

- By the reason of the analytical model and the in part not correct geometrically dimensions of slot and conductors in relation to the smartFEM model represent the results only rough approximation values which are afflicted with certain errors. For more accurate results are transient calculations required. This is not possible because the actually used FEM solver FEMAG only provides 2D-Simulation the electro-magnetic fields.

5 Simulation in Batch Mode

Batch files enable the sequential and parallel execution of several simulations The respective command lines for the simulations to be performed can be generated with a text editor. No specific order of parameters is prescribed.

5.1 Example of batch file with sequential execution of the simulations

```
ECHO OFF
ECHO smartFEM Batch Mode running, press "CTRL+C" to stop Batch Mode

REM smartFEM Batch Commands for smartFEM Release 2.7.0 and higher
REM
REM     [c:\...\]smartFEM.exe [help | /help | /?] [C:\...\]filename.mot -parameter
REM
REM     If smartFEM.exe and/or mot-file are stored in the folders assigned with
REM     smartFEM settings, then the complete paths are optional.
REM
REM     To display this file as help text [help|/help|/?] can be used.

REM Parameter:
REM
REM     "filename.mot" must be the first parameter, others can be in random sequence.

REM General options:
REM
REM     -log                Text file with log information will be saved with name "filename.log"
REM                         in the same folder as the mot-file .
REM
REM     -hidefemag          FEMAG windows will be closed and not shown.
REM
REM     -multi1|-multi2|... FEMAG will run on CPU (or Core) 1|2|...

REM Preprocessing options:
REM
REM     -top                        Load mot file with rotor and stator topology saved in mot-file.
REM
REM     -toprotor                   Load mot file with rotor topology saved in mot-file.
REM
REM     -topstator                  Load mot file with stator topology saved in mot-file.
REM
REM     -dxf                        Refreshing of both rotor and stator dxf-files is performed.
REM
REM     -dxfrotor                   Refreshing of rotor dxf-file is performed.
REM
REM     -dxfstator                  Refreshing of stator dxf-file is performed.
REM
REM     -nophaseshiftcalculation    no phase shift calculation will be performed

REM Calculation options:
REM
REM     -cogging            Cogging Torque + BEMF Calculation.
REM
REM     -nominal            Nominal Torque + Inductance Calculation.
```

```
REM
REM     -is400:-40            Current sine exitation is_rms=400A and Phase_shift_theta=-40°el
REM
REM     -iq306:-257           Current sine d/q excitation with iq_rms=306A and id_rms=-257A
REM
REM     -ldlq                 Calculation of user defined Ld/Lq sets.
REM
REM     -losses               Calculation of losses with smartFEM "Operational Speed" and
REM                           Speed Range defined inside "MotorParameters-Postprocessing-Losses",
REM                           loss results and loss data are saved.
REM
REM     -onlylossresults      Same as "-losses" but only loss results are saved.
REM
REM     -ls=100[:200:300:...]           Calculation of losses at different speed with named speed
REM                                     -ls=FirstSpeed and optional :SecondSpeed:ThirdSpeeed:etc.
REM
REM     -lsr=0:3000:11                  Calculation of losses at different speed with speed range
REM                                     -lsr=FirstSpeed:LastSpeed:NumberOfSteps.
REM
REM     -lextra filename.pex            Recalculation of extra losses defined in file.pex in Material folder
REM
REM     -force=0:180:2:stator           calculate forces from Nominal Torque + Inductance Calculation
REM                                     in "Motor Parameters - Post-Processing - Forces & Torque"
REM             -force=arc starts on:length of arc:number of segments:segments relative to stator|rotor

REM Calulation results options:
REM
REM     -export                         Run batch command and export results from mot-file
REM                                     to text file with name "filename.txt".
REM
REM     -export exportfilename.txt      Export of results to user defined "exportfilename.txt" file.
REM
REM     -export folderwithmotfiles      Run all batch command and export of results from each mot-file
REM                                     inside folder "...\folderwithmotfiles" to "FolderResults.txt"
REM                                     saved in the same folder.
REM
REM     -exportonlyoverview             Run batch command and export only overview table results
REM                                     of the mot-file to text file with name "filename.txt"

REM     Saving options:
REM
REM     -save [newmotfilename]          Open mot-file, refresh results and save motfile optional
REM                                               with new mot-file name.
REM
REM     -savenominaldone [newmotfilename]   as "-save" but with extra check whether nominal
REM                                               torque calculation is already performed. If not will
REM                                               it be performed.

REM     Remarks:
REM
REM     If the path of smartFEM.exe or mot-file contains " ", then must the path be included by
REM     quotation marks.

REM      e.g.:  D:\smartFEM\smartFEM.exe D:\elmoCAD\temp\Motor Design.mot >>
REM             D:\smartFEM\smartFEM.exe "D:\elmoCAD\temp\Motor Design.mot"
```

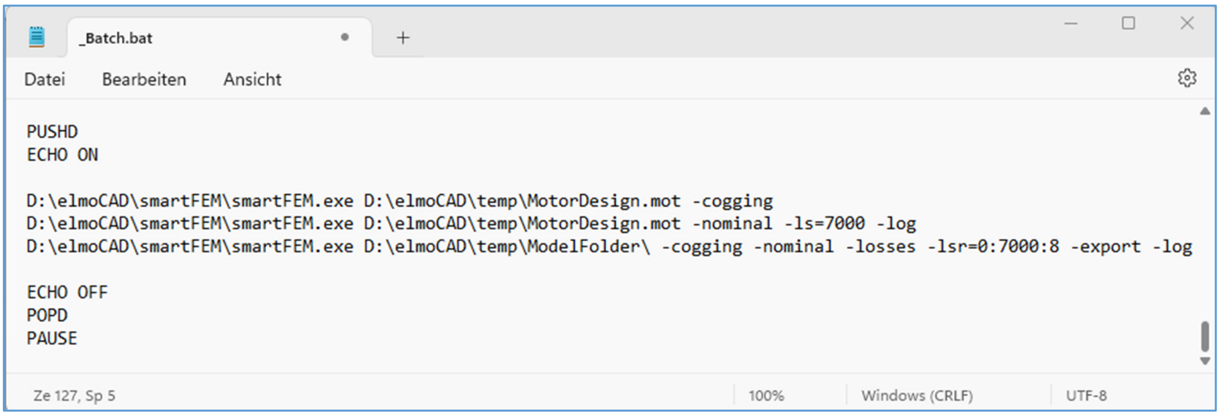
```
REM
REM Preprocessing will be automatically performed if required.
REM
REM Nominal Torque Calculation will be automatically performed if losses option is selected.
REM
REM ! For correct Nominal Torque results phase angle between BEMF and
REM ! Current has to be checked before calculations are performed, when
REM ! Sinusoidal or User Defined Current is selected.
REM
REM the results are stored in the selected file.
REM
REM for stop of calculations press "CTRL+C".

REM Examples:
REM Perform cogging calculation and write messages to log.
REM D:\elmoCAD\smartFEM\smartFEM.exe D:\elmoCAD\temp\MotorDesign.mot -cogging
REM
REM Perform nominal calculation and calculate losses for speed 7000 rpm
REM D:\elmoCAD\smartFEM\smartFEM.exe D:\elmoCAD\temp\MotorDesign.mot -nominal -ls=7000 -log
REM
REM Perform cogging and nominal calculation, calculate losses for speed 0rpm in steps of 1000rpm
REM until 7000rpm in steps of 1000rpm for all mot files inside modelfolder and export results to
REM "FolderResults.txt"
REM D:\elmoCAD\smartFEM\smartFEM.exe D:\elmoCAD\temp\modelfolder\ -cogging -nominal
REM -losses -lsr=0:7000:8 -export

PUSHD
ECHO ON

...\smartFEM.exe "...\ModelFolder\MotorDesign.mot" -cogging -log

ECHO OFF
POPD
PAUSE
```



```
PUSHD
ECHO ON

D:\elmoCAD\smartFEM\smartFEM.exe D:\elmoCAD\temp\MotorDesign.mot -cogging
D:\elmoCAD\smartFEM\smartFEM.exe D:\elmoCAD\temp\MotorDesign.mot -nominal -ls=7000 -log
D:\elmoCAD\smartFEM\smartFEM.exe D:\elmoCAD\temp\ModelFolder\ -cogging -nominal -losses -lsr=0:7000:8 -export -log

ECHO OFF
POPD
PAUSE
```


Fig. 173: Parameter settings - Sample batch file

If the mot files are stored in the *“temp”* directory specified with *"smartFEM Settings",* the path data can be omitted.

The "Preprocessing" function is performed automatically if necessary.

After starting the batch file, a DOS window is opened, in which the process is documented.

```
C:\WINDOWS\system32\cmd.exe

D:\Temp>echo off
Loading Motor Design File: D:\Temp\test1.mot

Running Preprocessing
FEMAG Preprocessing Successfully Finished
FEMAG is Calculating Cogging Torque With Complete Number of Steps
Motor Desing Saved in File: D:\Temp\test1.mot
Calculation of Cogging Torque And BEMF with FEMAG Successfully Finished
Motor Desing Saved in File: D:\Temp\test1.mot
Loading Motor Design File: D:\Temp\test2.mot
```

Fig. 174: Parameter settings - Sample Batch log file

The DOS window is closed after all command lines have been executed, unless the DOS command "Pause" is used in the last command line. The messages sent by smartFEM to the Dos window are stored in a log file if the -log parameter is used.

5.2 Exporting result data to a text file

The result data displayed in *"Results"* are exported to a text file using the batch command *"-export"*, which in turn can be opened in a spreadsheet for further individual calculations. There are two options for export:

- Export the results of one machine model (* .mot file)

  Command: c: \ ... \ MotorVarianten \MotorDesign1.mot -export c: \ ... \ MotorDesign1_Results.txt

  If no path is specified, the export file is stored in the directory where the mot file is located.

- Export the results of all machine models inside the folder Command:

  Command: -export c: \ ... \ Motorvarianten Exported_Results.txt

  In this example, the results of all machine models located in the c: \ ... \ MotorVarianten directory are exported to Exported_Results.txt.

	A	B	C	D	E	F	G	H	I	J	K
1	**Export\Motorvariante_1.mot**										
2	Cogging Torque			Flux per Phase Without Applied Current					BEMF per Phase		
3	max[Nm]	0,0069		max[Vs]	0,2175558	0,2175558	0,2175558		max[V]	140,183844	140,
4	min[Nm]	-0,0071		min[Vs]	-0,21755811	-0,21755811	-0,21755811		min[V]	-140,121843	-140,1
5	AVG[Nm]	-4,48E-05		AVG[Vs]	7,52E-08	3,90E-08	-5,89E-09		AVG[V]	-3,95E-16	2,:
6	RMS[Nm]	0,00413039		RMS[Vs]	0,15234267	0,15234272	0,15234271		RMS[V]	95,8105534	95,81
7	Ripple[%]	-31274,4476		Ripple[%]					Ripple[%]		
8	α [°mech]	Tcogg[Nm]		α [°mech]	Flux_U[Vs]	Flux_V[Vs]	Flux_W[Vs]		α [°mech]	BEMF_U[V]	BEMF_
9	0	2,20E-05		0	0,18473868	-1,20E-05	-0,18472713		0	-70,0141698	139,7
10	0,5	0,0009475		0,5	0,18277686	0,00386484	-0,18664296		0,5	-71,3753281	139,5
11	1	0,001885		1	0,18078018	0,00774081	-0,18852372		1	-72,6993401	139,4
12	1,5	0,002775		1,5	0,17873583	0,01161468	-0,1903545		1,5	-74,0547646	139,5
13	2	0,003625		2	0,17666292	0,01548183	-0,19214937		2	-75,504705	139,6
744											
745											
746	**Export\Motorvariante_3.mot**										
747	Torque With Applied Current			Cogging Torque			Current per Phase				
748	max[Nm]	1,76		max[Nm]	0,00686		max[A]	2,99999998	2,99999998	2,99999998	
749	min[Nm]	1,49		min[Nm]	-0,0069		min[A]	-2,99271729	-2,99271721	-2,99271736	
750	AVG[Nm]	1,60333333		AVG[Nm]	-1,41E-05		AVG[A]	-9,18E-16	1,97E-17	1,24E-15	
751	RMS[Nm]	1,60602823		RMS[Nm]	0,00412153		RMS[A]	2,12132034	2,12132034	2,12132034	
752	Ripple[%]	16,8399168		Ripple[%]	-97419,7609		Ripple[%]				
753	α [°mech]	T[Nm]		α [°mech]	Tcogg[Nm]		α [°mech]	i_U[A]	i_V[A]	i_W[A]	
754	0	1,51		0	1,61E-05		0	-1,49968762	2,99999998	-1,50031326	
755	4	1,49		4	0,00637		4	-1,84670018	2,97085422	-1,12415517	
756	8	1,52		8	0,00465		8	-2,15776883	2,88388416	-0,72611668	
757	12	1,63		12	-0,00195		12	-2,42683896	2,74078258	-0,31394515	
758	16	1,74		16	0,00125		16	-2,64867343	2,54433479	0,10433697	

Fig. 175: Batch simulation- Exported result file

5.3 Parallel Computing

If several processor cores are present on a PC, different calculation runs can be distributed to the existing processor cores for parallel processing. Thus, a considerable amount of computation time can be saved, especially when, for example, a Nominal Torque Calculation takes several hours.

As exemplary templates, there are two batch files in the batch directory ...\smartFEM\.

5.3.1 _Batch_MultiRun.bat

Here, as in Batch.bat, the calculations are determined and it is defined on which processor core they are to run:

```
echo off
rem smartFEM calculations with different mot-files, whereby parallel
rem computing on the existing CPU's or Core's of the PC is performed
rem
rem This batch must be started via _Batch_MultiRunStart.bat
rem
.
.
.
rem Nominal Torque calculation will be automatically performed before
rem calculations if necessary.
.
.
.
rem example:
rem     motfile-1 ... motfile-4 are running parallel on 4 cores
rem     motfile-5 ... motfile-8 are running parallel on 4 cores
rem
rem     motfile-5 is running in sequence after motfile-1 is finished
rem     motfile-6          "          motfile-2    "
rem     motfile-7          "          motfile-3    "
rem     motfile-8          "          motfile-4    "

set exepath=C:\Users\elmoCAD\smartFEM
set motpath=C:\Users\elmoCAD\Temp

set motfile-1=MotorDesignTemp_1.mot
set motfile-2=MotorDesignTemp_2.mot
set motfile-3=MotorDesignTemp_3.mot
set motfile-4=

set motfile-5=
set motfile-6=
set motfile-7=
set motfile-8=

if %1 == -multi1 goto :CPU1
if %1 == -multi2 goto :CPU2
if %1 == -multi3 goto :CPU3
if %1 == -multi4 goto :CPU4

Goto :End
```

```
rem ***********
:CPU1
echo mot-file running on CPU1
echo on
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-1% %1 -log -hidefemag
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-5% %1 -log -hidefemag

Goto End

rem ***********
:CPU2
echo mot-file running on CPU2
echo on
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-2% %1 -log -hidefemag
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-6% %1 -log -hidefemag

Goto End

rem ***********
:CPU3
echo mot-file running on CPU3
echo on
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-3% %1 -log -hidefemag
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-7% %1 -log -hidefemag

Goto End

rem ***********
:CPU4
echo mot-file running on CPU4
echo on
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-4% %1 -log -hidefemag
%exepath%\smartFEM.exe -nominal %motpath%\%motfile-8% %1 -log -hidefemag

Goto End

rem ***********
:End
Pause
```

The batch must be adapted according to the number of processor cores available on the respective PC.

This batch must be started by batch file *"_Batch_StartMultiRun.bat"*

5.3.2 _Batch_StartMultiRun.bat

Batch calls up the previously described *_Batch_MultiRun.bat* batch several times according to the defined number of processor cores, without waiting for the complete execution. The respective processor cores to be used are transferred as parameters.

The batch must also be adapted with regard to the available number of processor cores of the PC.

```
echo off
rem smartFEM calculations with different mot-files, whereby parallel
rem computing on the existing CPU's or Core's of the PC is performed
rem
rem This batch will start _Batch_MultiRunStart.bat where mot.files and
rem CPU's have to be assigned
rem

pushd

start /min C:\Users\elmoCAD\Temp\_Batch_MultiRun.bat -multi1
start /min C:\Users\elmoCAD\Temp\_Batch_MultiRun.bat -multi2
start /min C:\Users\elmoCAD\Temp\_Batch_MultiRun.bat -multi3
start /min C:\Users\elmoCAD\Temp\_Batch_MultiRun.bat -multi4

popd
exit
```

5.3.3 smartFEM

smartFEM can also be started with the following command on a specific or several processor cores:

smartFEM.exe -multi # (# = number of the processor core)

With this is also parallel computing simulation possible.

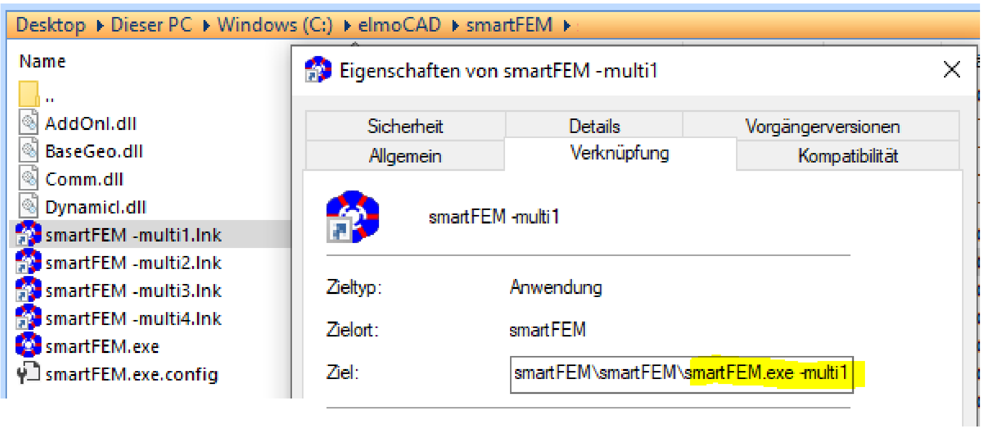


Fig. 176: Parallel Computing - Assignment of processor cores

6 Special modules

6.1 PM - Magnetization - Inner rotor

The motor type *"PM - Magnetization - Inner Rotor"* is used to simulate the magnetization and the remanence of the magnets of a rotor. They are equipped with non - magnetized magnetic material and subsequently magnetized by means of a magnetizing coil. Two methods are implemented for the calculation of remanence: a) FEM calculation with FEMAG and b) analytical calculation, which can be used if the iron core of the magnetization coil is not made of sheet material and very high eddy currents occur.

Any rotor that can be selected in smartFEM can be used for internal rotors. In the first step, the rotor is modeled as usual with the corresponding topology.

6.1.1 Rotor and magnetizing coil

For the modeling of the magnetizing coil, a corresponding topology of the stator *"MD ... .top"* is included in smartFEM. The number of slots corresponds automatically to the number of magnets defined in the rotor.

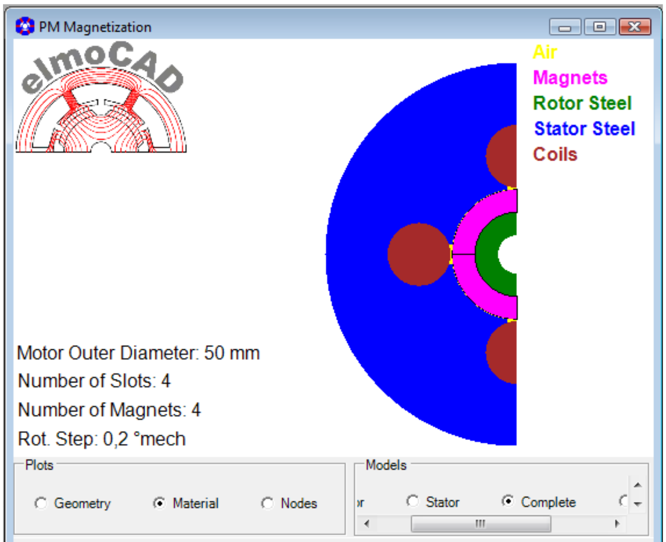


Fig. 177: PM magnetization - Example of Magnetizing device (coil and rotor)

A material file, which contains a new curve of the "Soft Iron" magnetic material in the first quadrant as the B (H) curve, must be selected as the magnetic material in "Material".

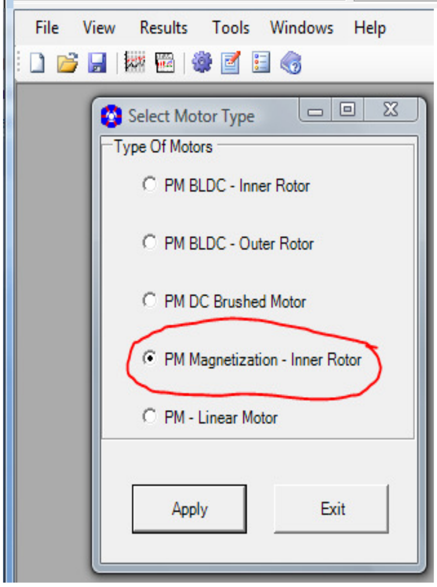


Fig. 178: PM magnetization - Initialization

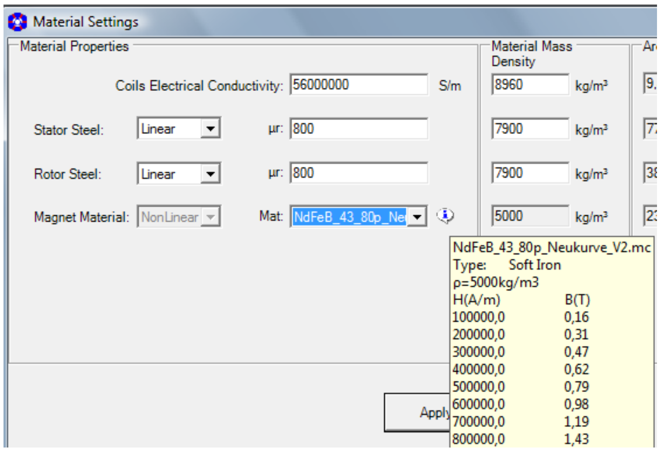


Fig. 179: PM-Magnetization - Magnet Material

The data for a corresponding new curve B(H) can be recorded using the material Explorer or PowerCore® Explorer contained in smartFEM.

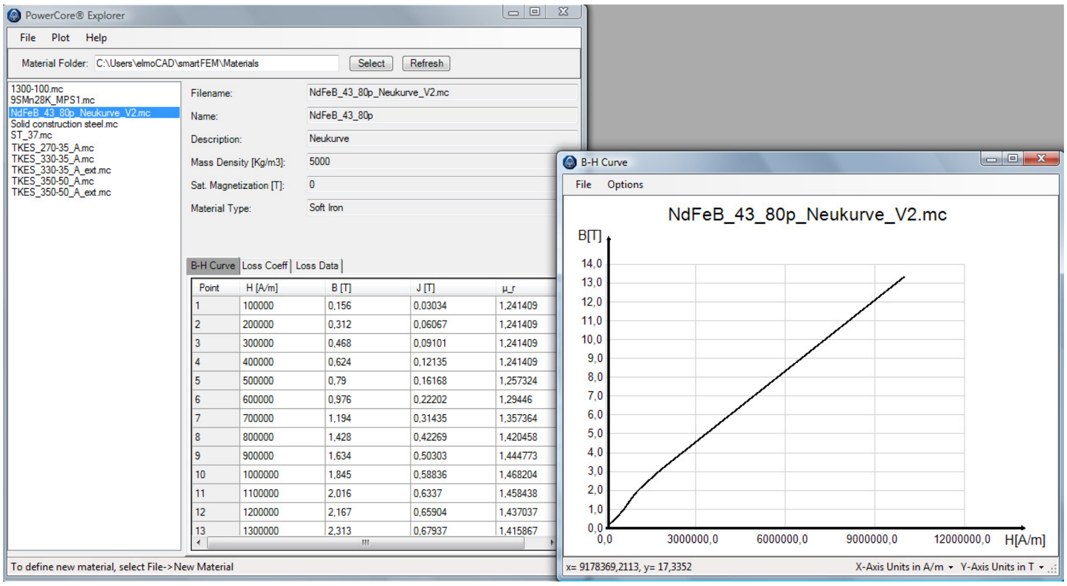


Fig. 180: PM magnetization - new curve of a neodymium iron boron material

In *"Motor Parameters"* the data for magnetization simulation are specified subsequently:

- Min. Rotation Step: minimum step angle for the rotation of the rotor
- Convergence Accuracy: abbreviation criterion for FEMAG interactions
- Magnetization Current: amplitude of the magnetization current
- Remanence by: FEMAG: the remanence induction is determined by FEMAG
  MEXFile: the remanence is specified in a *"Magnetization Activation"* table.
- Nominal Remanence Brem: nominal remanence at the chosen current
- Anisotropy Mode: isotropic or anisotropic

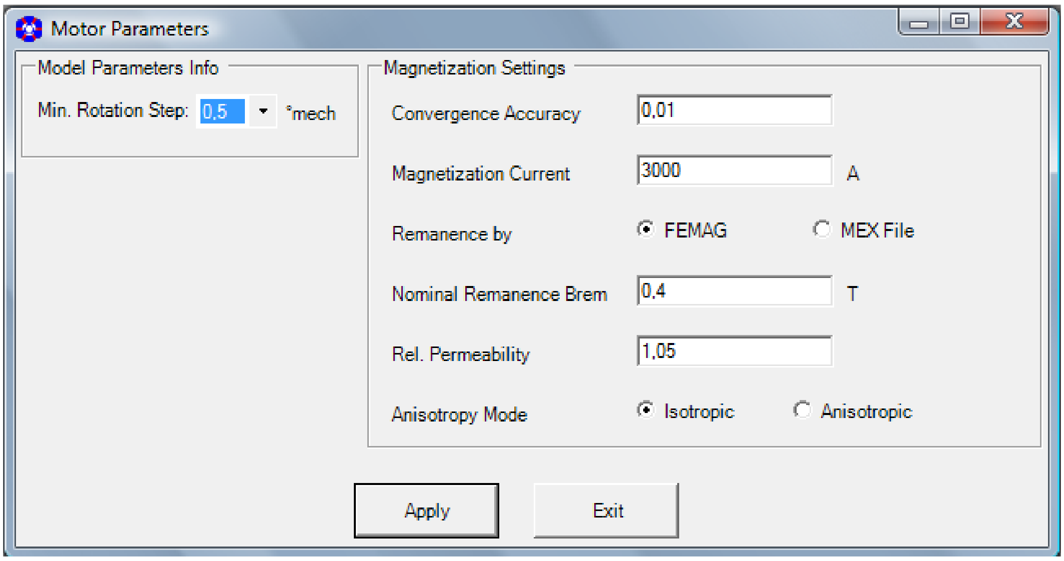


Fig. 181: PM magnetization - Motor parameters

The simulation is started by pressing the *"Run Magnetization"* button,.

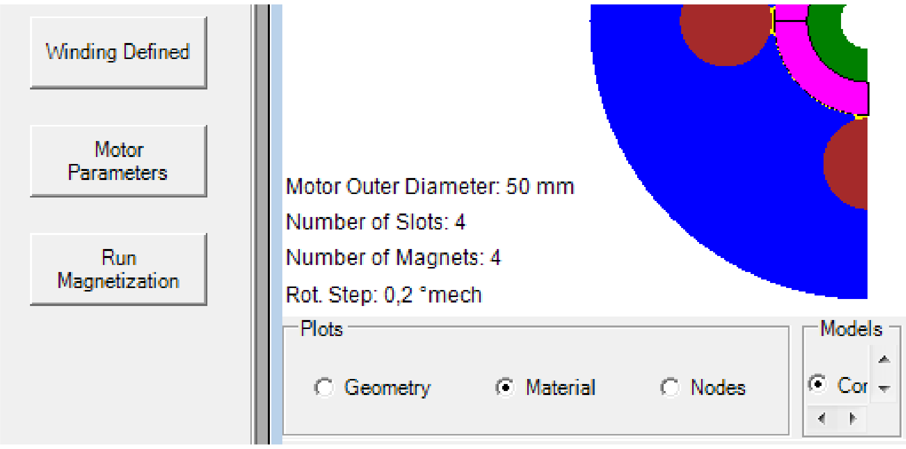


Fig. 182: PM magnetization - Start of the simulation with "Run Magnetization"

The simulation results can be displayed and edited as usual in *"Plots"*, e.g.:

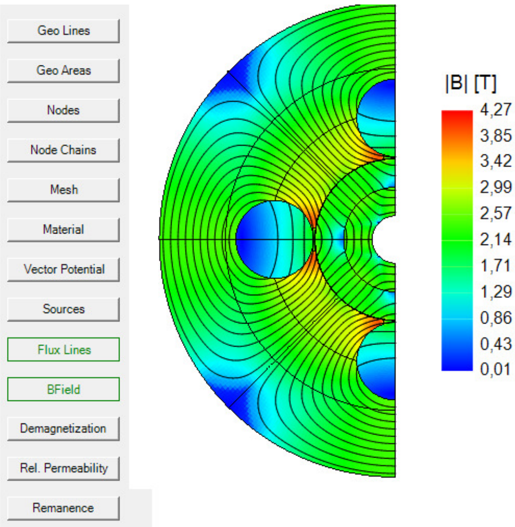


Fig. 183: PM magnetization - Induction distribution

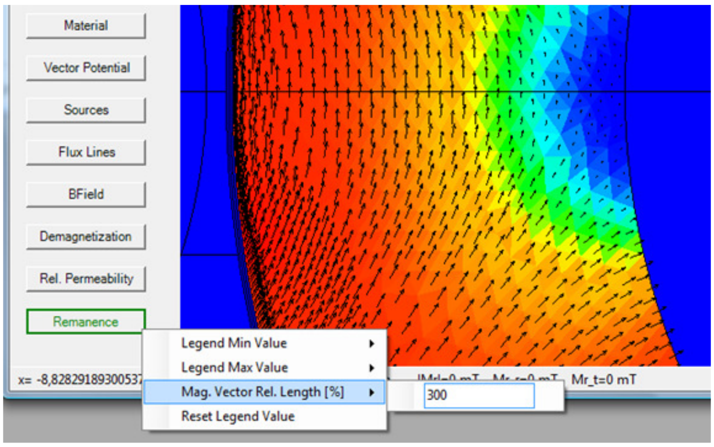


Fig. 184: PM magnetization - Remanence in the ring magnet (zoomed)

The induction, which is actually reached in air at a certain distance from the magnetic surface, can be calculated and displayed as a * .mot file in a second step after saving the simulation model.

6.1.2 Calculation and evaluation of air gap induction and remanence

To assess the magnetization, the air gap induction of pre-magnetized rotors can be calculated and displayed using a "measuring ring" stator topology (Messring). For this purpose, the previously stored simulation model of the pre-magnetized rotor is loaded in the rotor topology of a new model through the dialog *"File - Load Magnetization".*

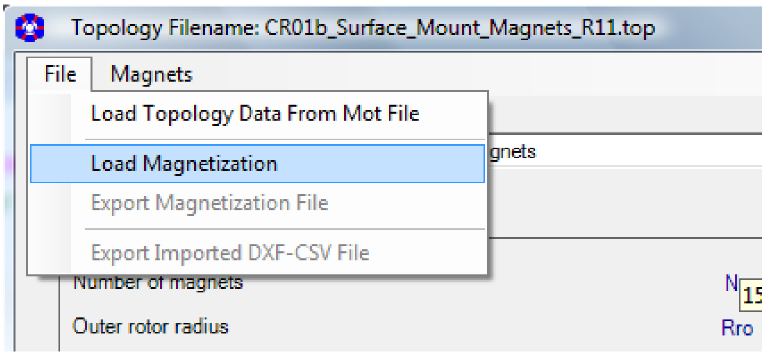


Fig. 185: PM magnetization - Charging a pre-magnetized rotor model

Additionally, the stator topology *"CS01_Measure_Ring_Magnetization"* is selected and correspondingly parameterized so that it corresponds to a device for measuring the remanence induction of the magnets. The distance between the measuring probe and the magnet must be equal to the distance of the middle air gap, so that measurement and simulation results can be compared. If the measurement is carried out in air, the thickness of the stator ring should be a multiple of the rotor radius in order not to limit the field propagation in the air by boundary conditions of a ring that is too narrow.

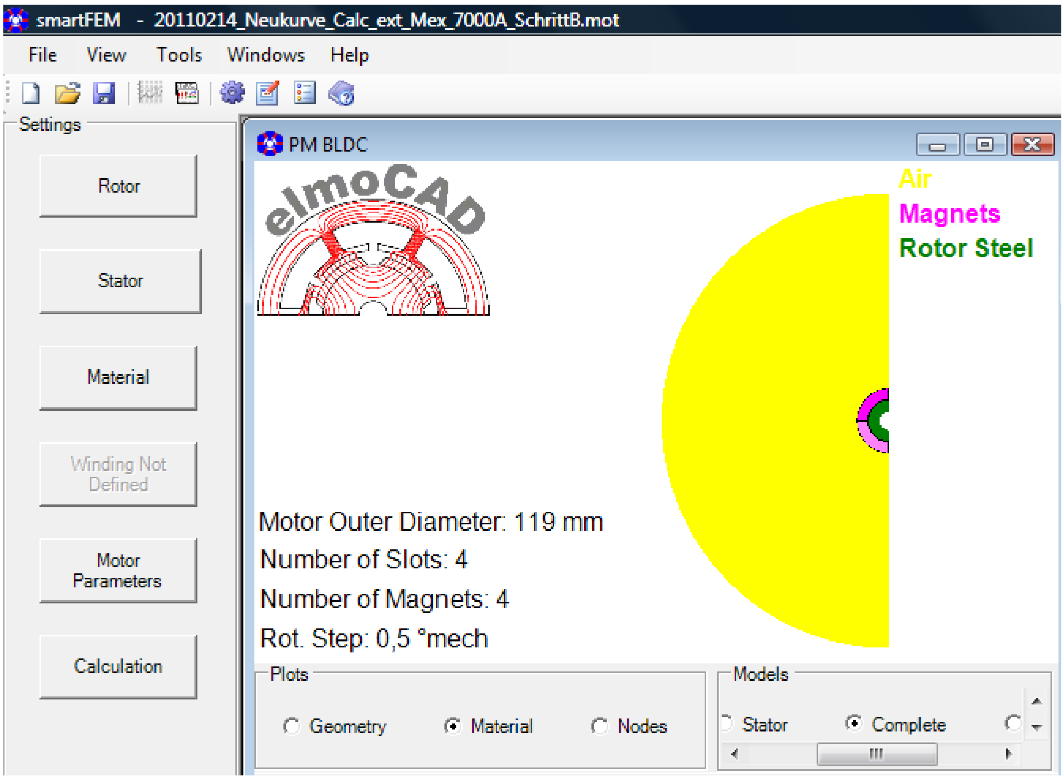


Fig. 186: PM magnetization model for simulating the induction on the rotor surface

The simulation is started with the *"Calculation"* button. Additionally, the results (like for example the calculated air gap induction) can be calculated both with the *"BField"* plot as well as with *"Postprocessing - Forces and Torque - Calculate"* in *"Motor Parameters".*

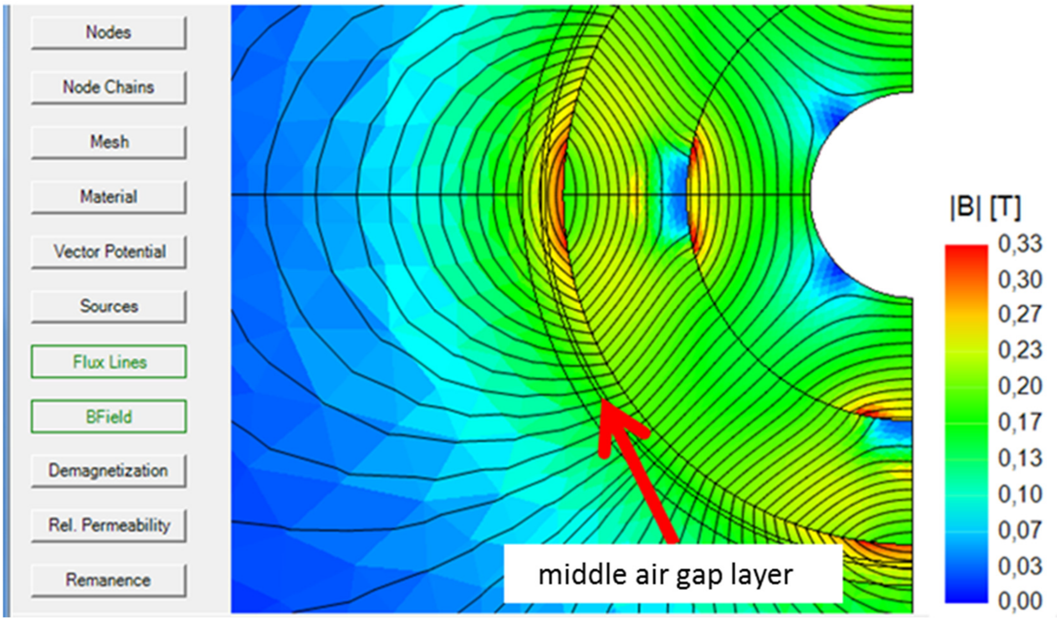


Fig. 187: PM magnetization - Induction distribution plot

The remanence induction of the magnets corresponds to the evaluation shown in Fig.112.

The values of the air gap induction (radial and tangential) can be copied via the buffer to a spreadsheet for further evaluations through *"Options - Copy Airgap B-Field Values to Clipboard".*

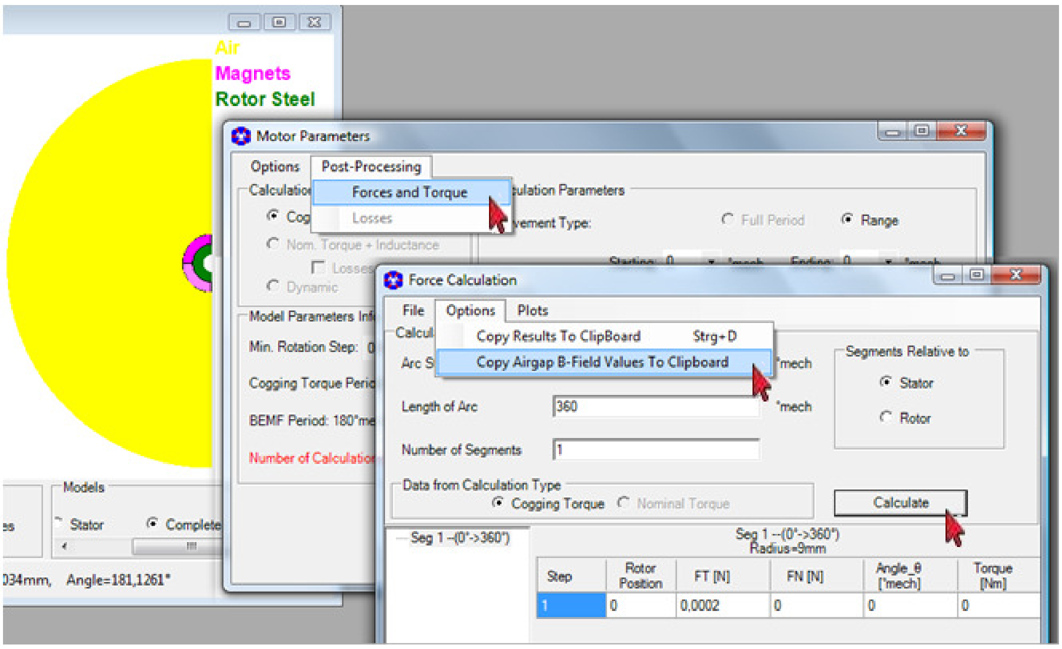


Fig. 188: PM magnetization - Calculating air gap induction by Post-Processing

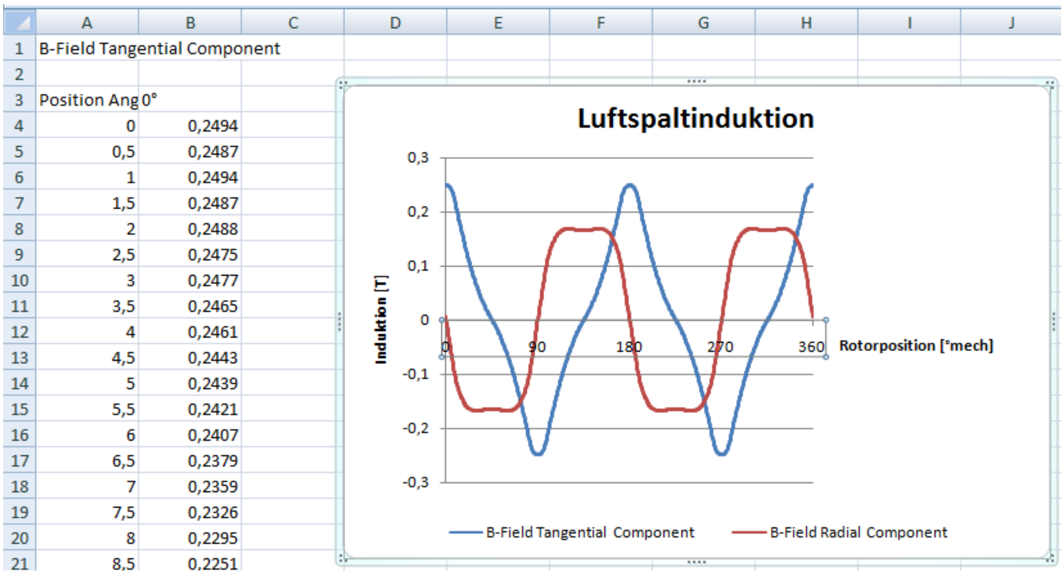


Fig. 189: PM magnetization - Air gap induction evaluated in spreadsheets

6.1.3 Use of pre-magnetized rotors in motor models

The simulation models of pre-magnetized rotors saved as * .mot file can be used in motor models. For this purpose, the corresponding rotor model is loaded in the rotor topology via the dialog *"Options - Load Magnetization"* (see Fig. 113).

The geometry parameters of the rotor and the magnetic material can no longer be changed. If necessary, this must be done as in Tx. 6.1.1 and the rotor can be magnetized again.

All other functions for the simulation of the motor model with pre-magnetized rotor are available as usual.

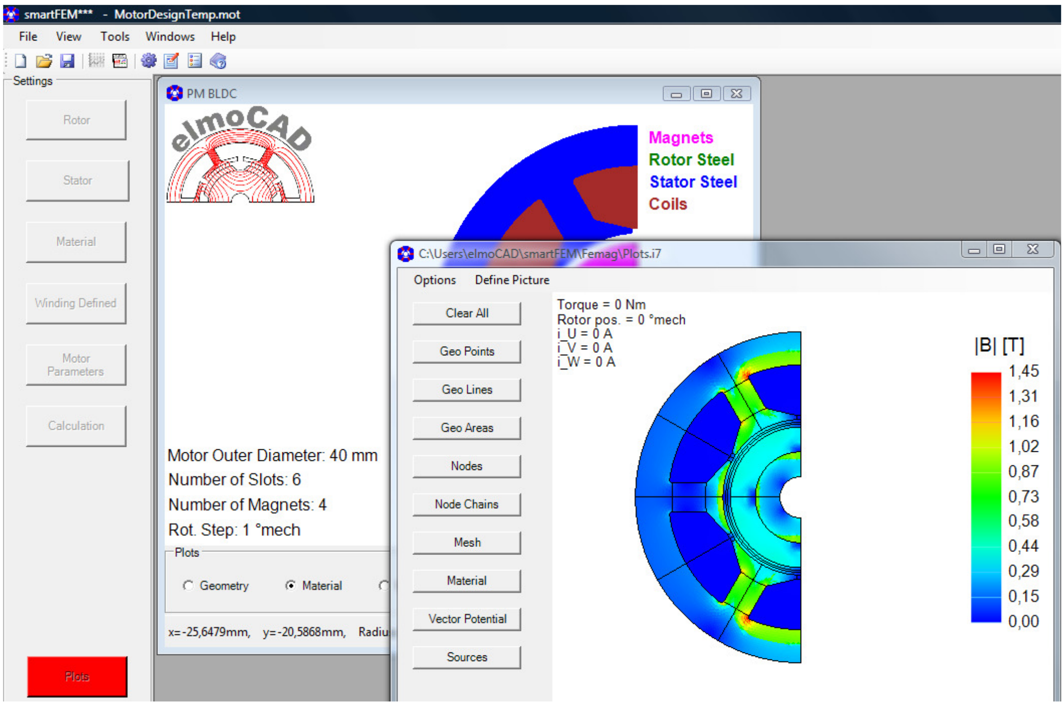


Fig. 190: PM magnetization - Motor model with biased rotor

6.2 PM DC Brushed Motor

The creation of a simulation model PM DC Brushed Motor is initialized via *"File - New - Select Motor Type"*. The topologies correspond to those of the outer rotor motors. At least one PM-BDLC outer rotor topology (* .top) must be stored in the topology directory in order for menu item *"PM DC Brushed Motor"* to be displayed.

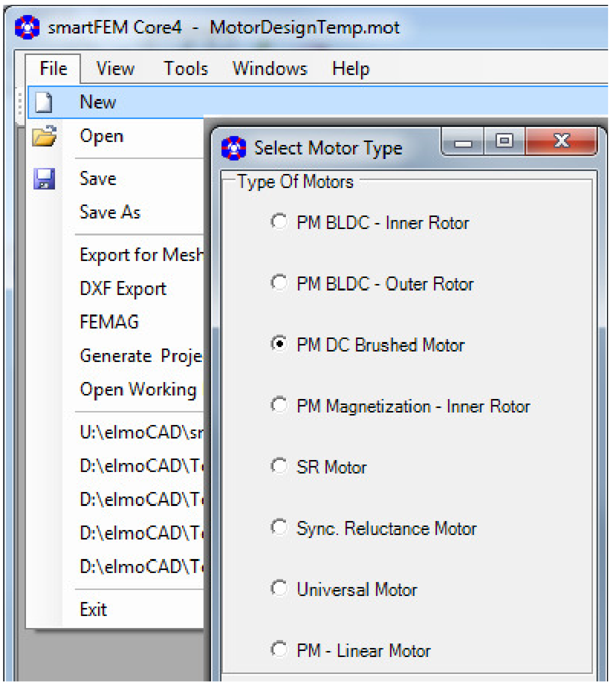


Fig. 191: PM DC Brushed Motor initialization

All outer rotors and stators can be selected.

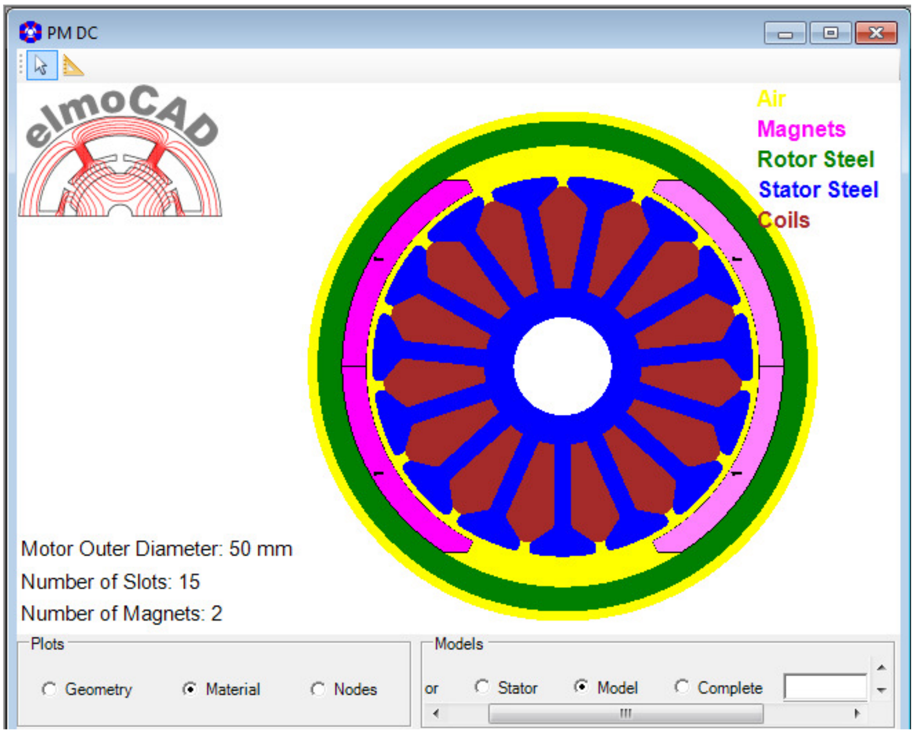


Fig. 192: PM DC Brushed Motor - Example of a geometry model

The Winding Editor generates automatically a winding scheme which also can be edited by the users and it provides the possibility to create different Phasor Diagrams

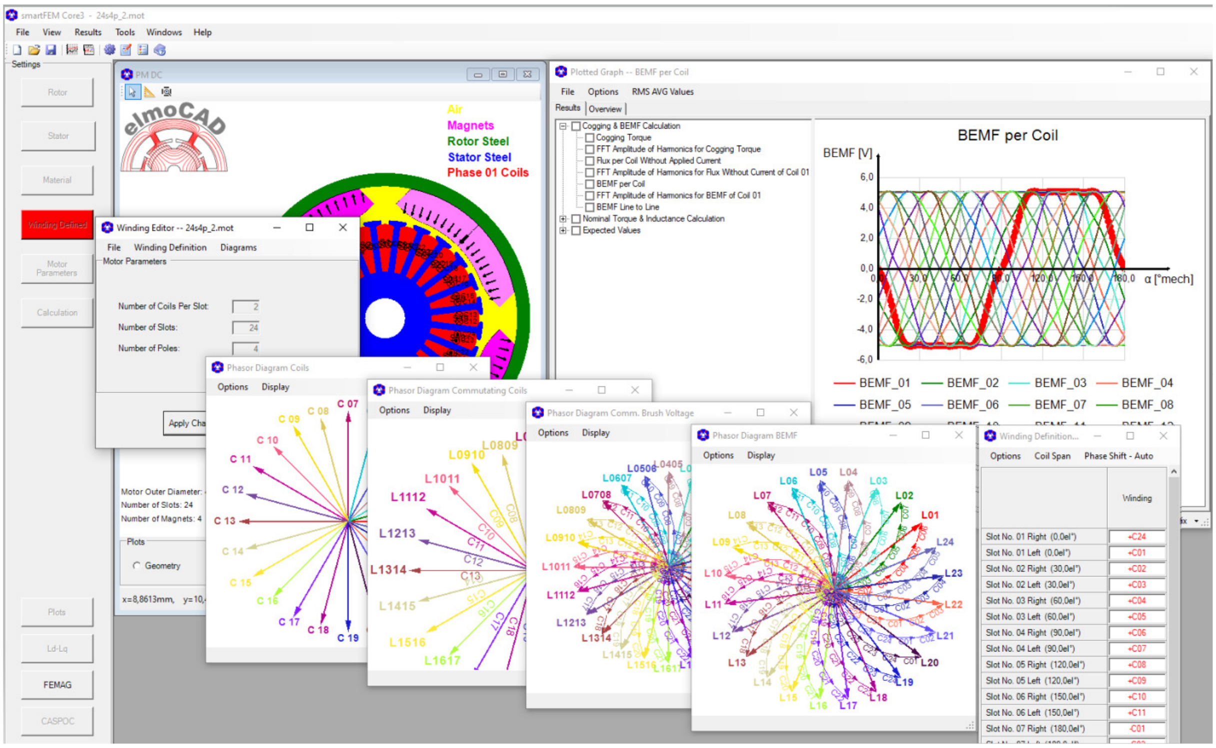


Fig. 193: PM DC Brushed Motor - Winding Scheme and Phasor Diagrams

Winding Scheme, coil parameter, line to line definition and commutation sequence can be adapted manually.

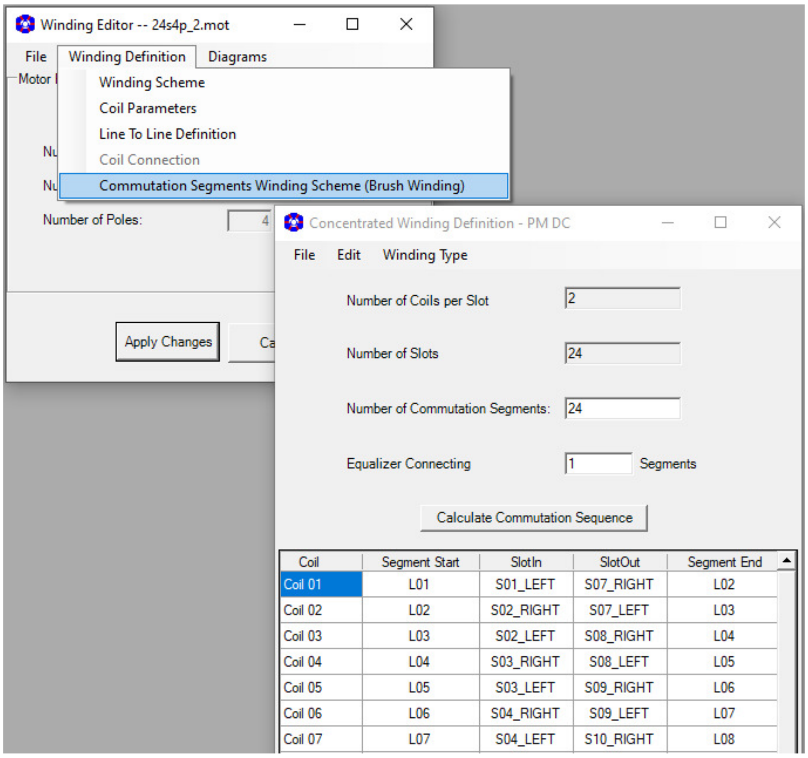


Fig. 194: PM DC Brushed Motor - Commutation Sequence

The position of the brushes to the segments of the commutator are defined in *"MotorParameters".*

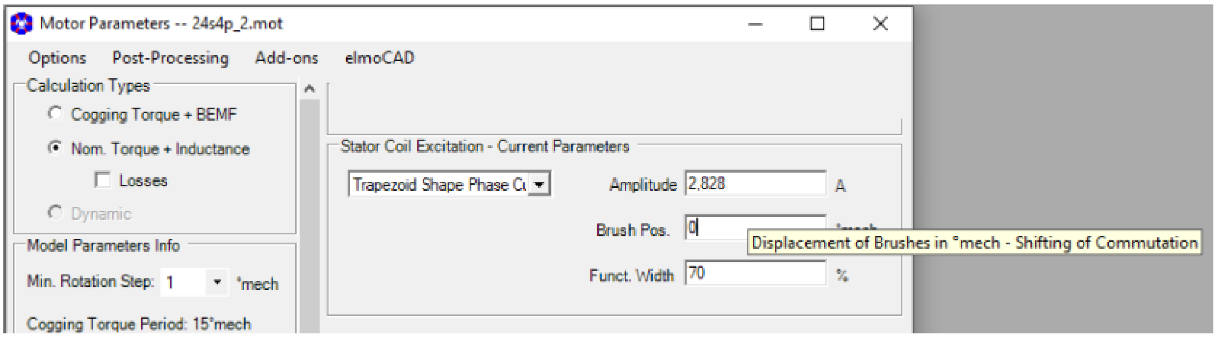


Fig. 195: PM DC Brushed Motor - Brush position

6.3 Synchronous Motor (external excited)

The simulation model of an externally excited "*Synchronous Motor*" is initialized by the menu sequence „*File - New - Select Motor Type*". The stator topologies which can be selected are the same as for *"PM BLDC - Inner Rotor"* motors. At least must one rotor topology (*.top file) with external excitation be saved in the smartFEM topology folder so that the menu item *„Synchronous Motor"* is displayed.

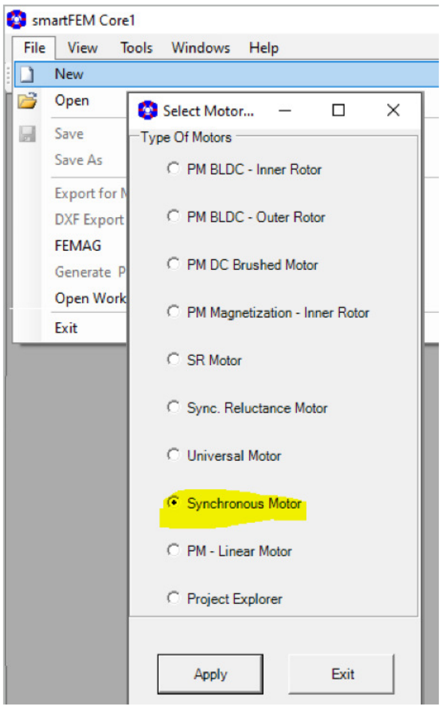


Fig. 197: Synchronous Motor - Initialization

Fig. 196: Synchronous Motor with 4 poles

The conductivity of the rotor coils can be defined in „*Material*". Default value is 56000000 S/m.

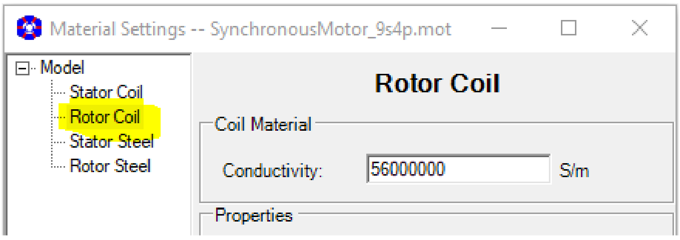
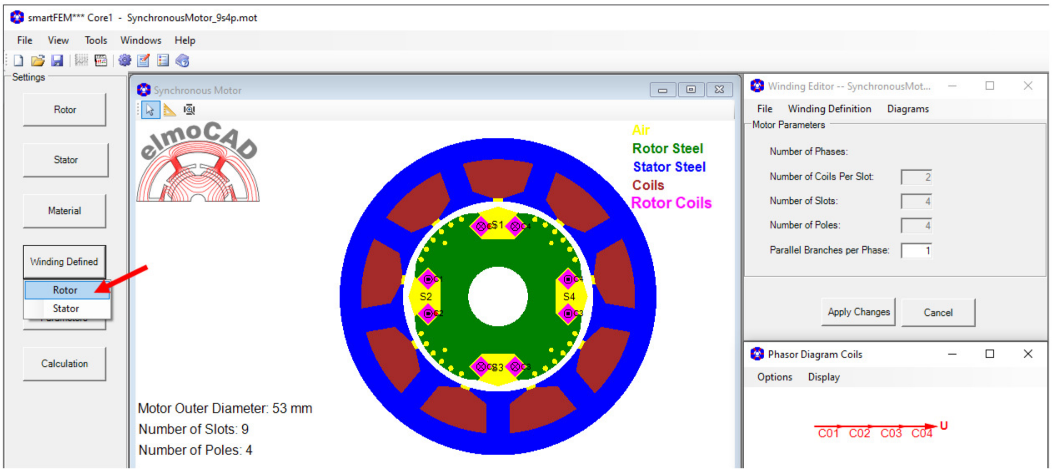


Fig. 198: Synchronous Motor - Material Settings

The parameter of rotor and stator windings can be defined separately.

Fig. 199: Synchronous Motor - Windings



The excitation of the rotor is defined as constant current in „*MotorParameters*".

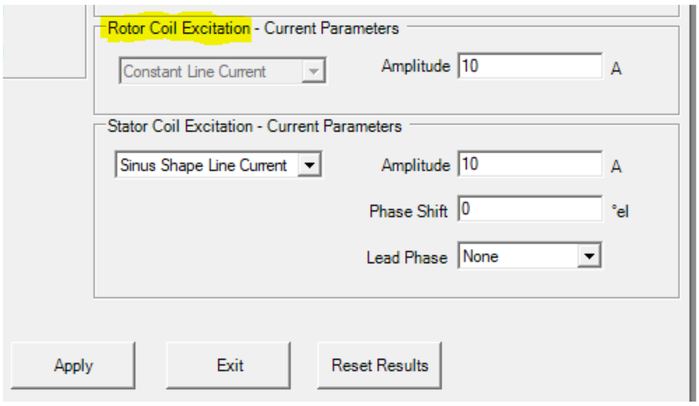


Fig. 200: Synchonous Motor - Rotor Coil Excitation

All calculations as of *"PM BLDC ..."* motors except *„Cogging Torque + BEMF Calculation"* can be performed.

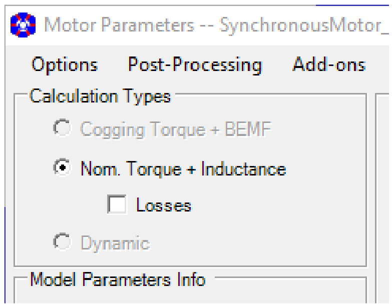


Fig. 201: Syncronous Motor - Calculation Types

The BEMF has to be calculated separately in an extra smartFEM model by *„Nominal Torque + Inductance Calculation"* with rotor current Ir > 0 A and stator current Is = 0 A .

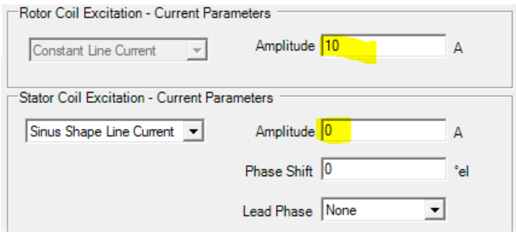


Fig. 202: Synchronous Motor - Coils Excitation

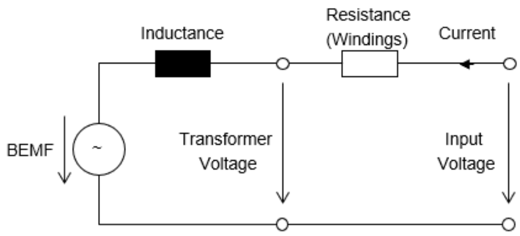


Fig. 203: Synchronous Motor - Equivalent Circuit

The BEMF is equal to the result of the *„Transformer Voltage"*.

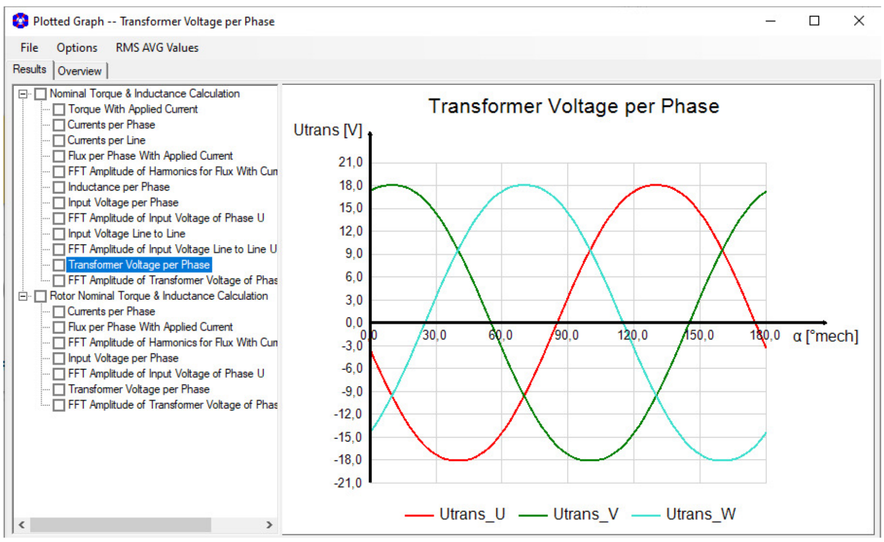


Fig. 204: Synchronous Motor - BEMF equivalent

The calculation of the BEMF is not mandotary. All results of the „*Nominal Torque + Inductance Calculation*“ are available in *“Results”* after the calculation was performed.

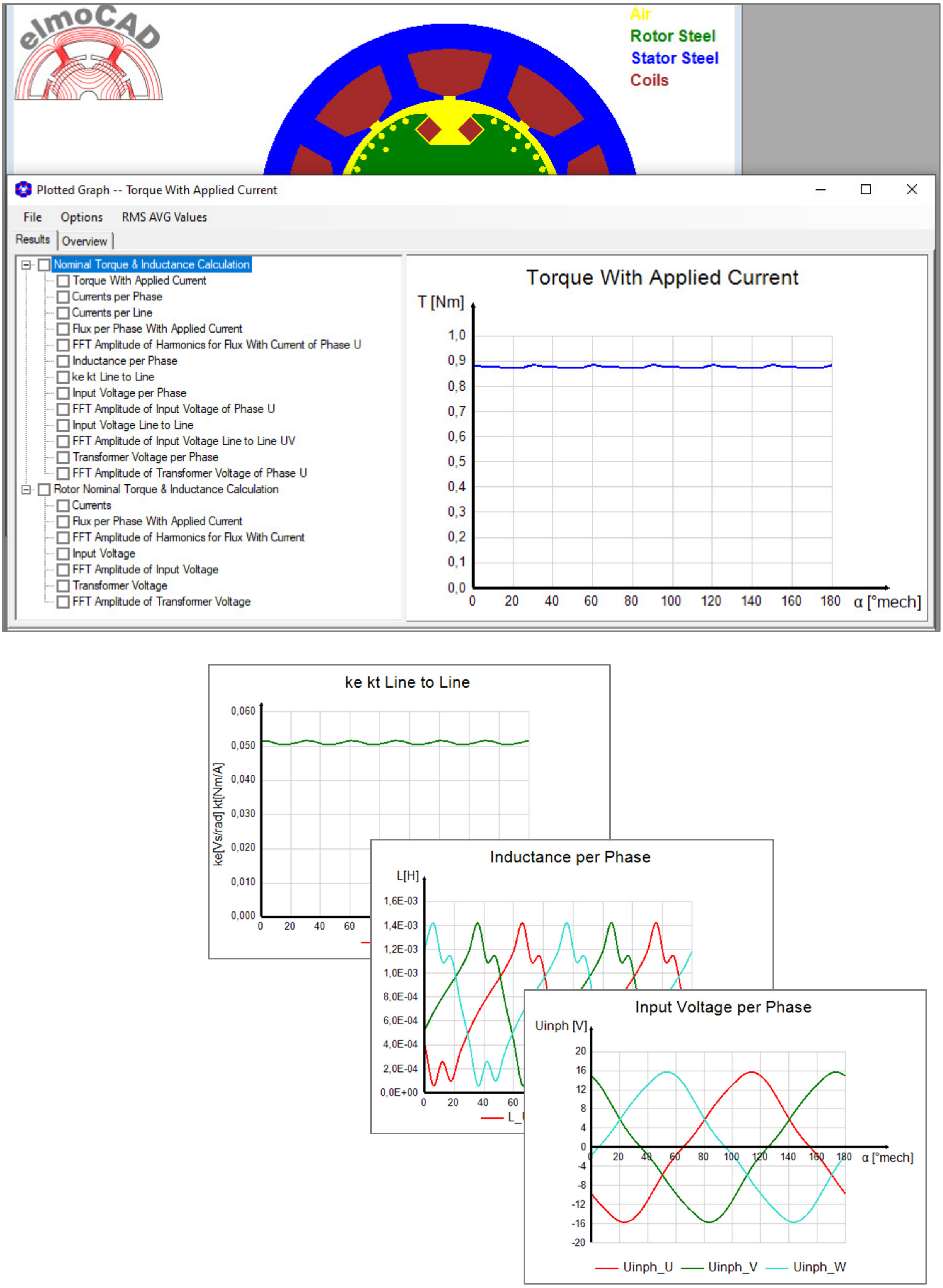


Fig. 205: Synchronous Motor - Results of "Nominal Torque ... Calculation"

6.3.1 Synchronous Motor - Field Plots

In *"Plots"* is the rotor current entered in column "i_r" manually or inserted by menu sequence *„Data - Paste Currents to set No. # - source"*.

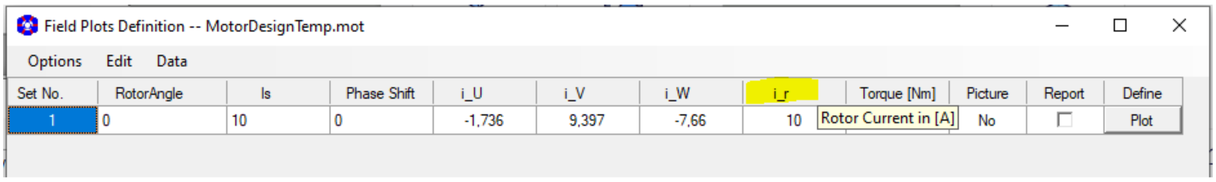


Fig. 206: Synchronous Motor - Rotor Coil Excitation in "Plots"

The current densities are set in rotor and stator winding related to the winding scheme.

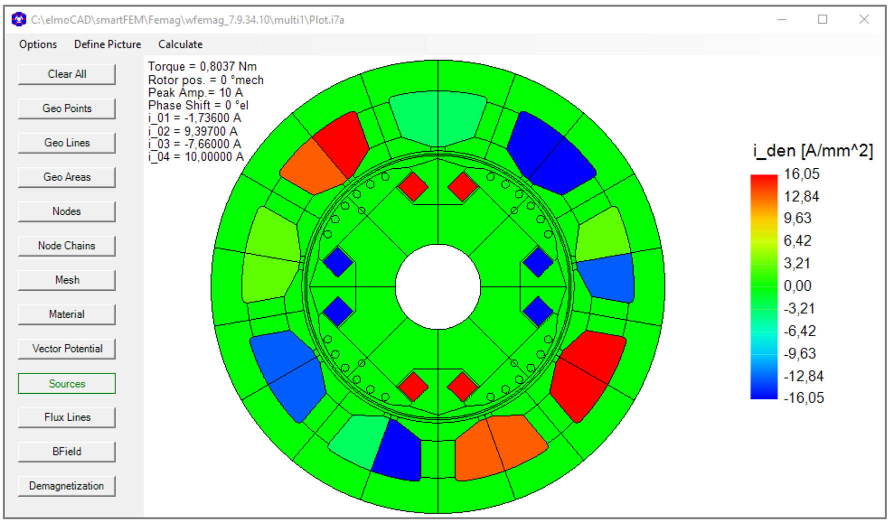


Fig. 207: Synchronous Motor - Plots - Excitation Sources

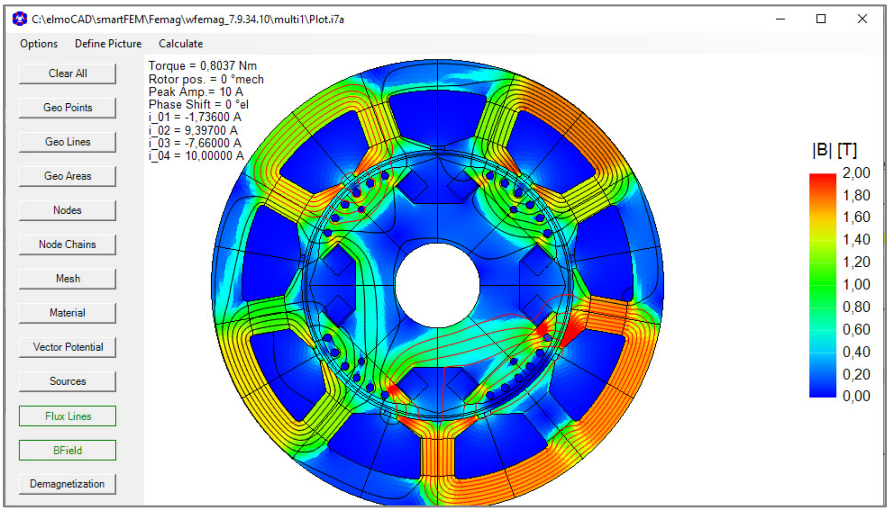


Fig. 208: Syncvhronous Motor - Plots - Flux Densities and Flux Lines

6.4 Switched Reluctance Motor

The creation of a SR motor model simulation is initialized via *"File - New - Select Motor Type"*. At least one SR rotor topology (* .top) must be stored in the topology directory in order to display the *„SR - Motor“* menu item.

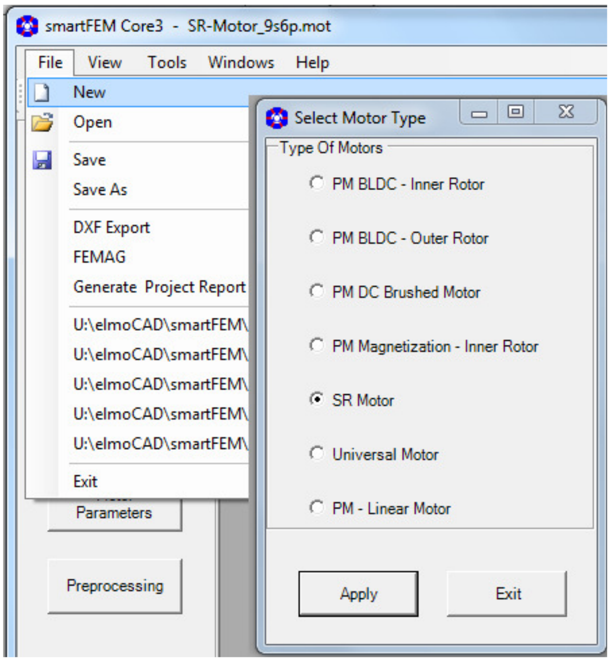


Fig. 209: SR motor initialization

The rotor topology of an SR motor differs from that of a PM motor in that only rotors without magnets can be selected.

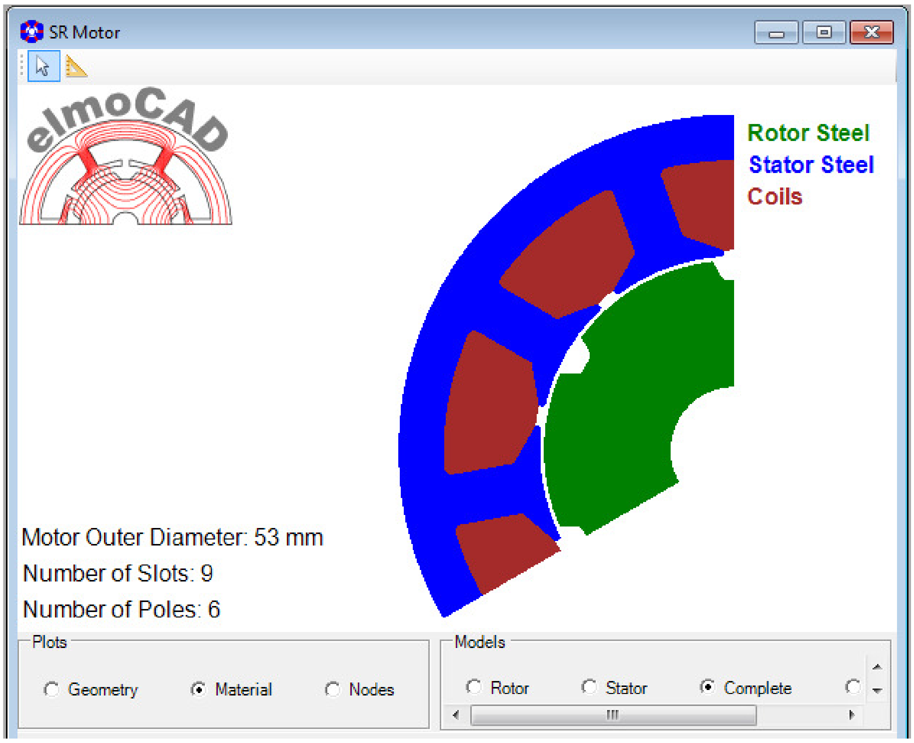


Fig. 210: SR motor - Example of a geometry model

In *„Motor Parameters“*, the current can be specified as *"User Defined Phase Current"*.

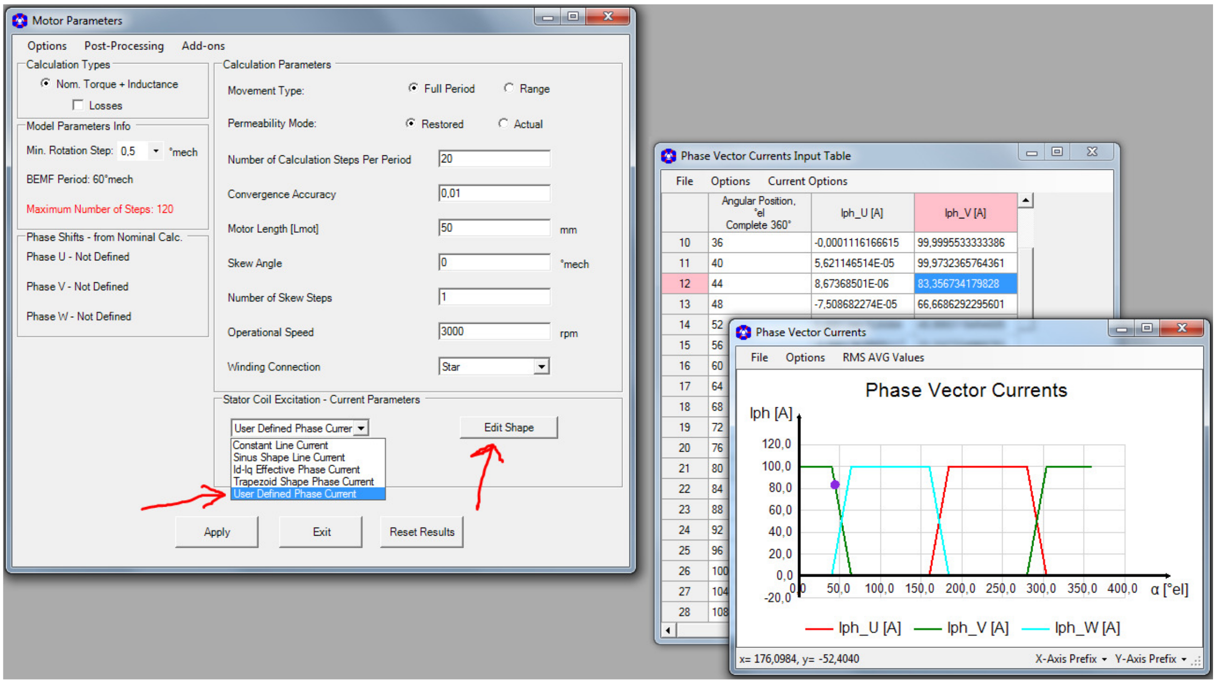


Fig. 211: SR motor - User Defined Phase Current

Because the phase positions of the windings of a SR motor cannot be determined from motor topology and winding scheme, a torque calculation with a constant current is performed for each phase. The maxima of torques then correspond to the respective phase position. To perform this calculation, the *"ReCaculate Phase Current"* button must be clicked in *"Motor Parameters"*. Subsequently, at the next *"Nom. Torque + Inductance"* calculation, the phase positions are calculated before and displayed in *"Motor Parameters"*.

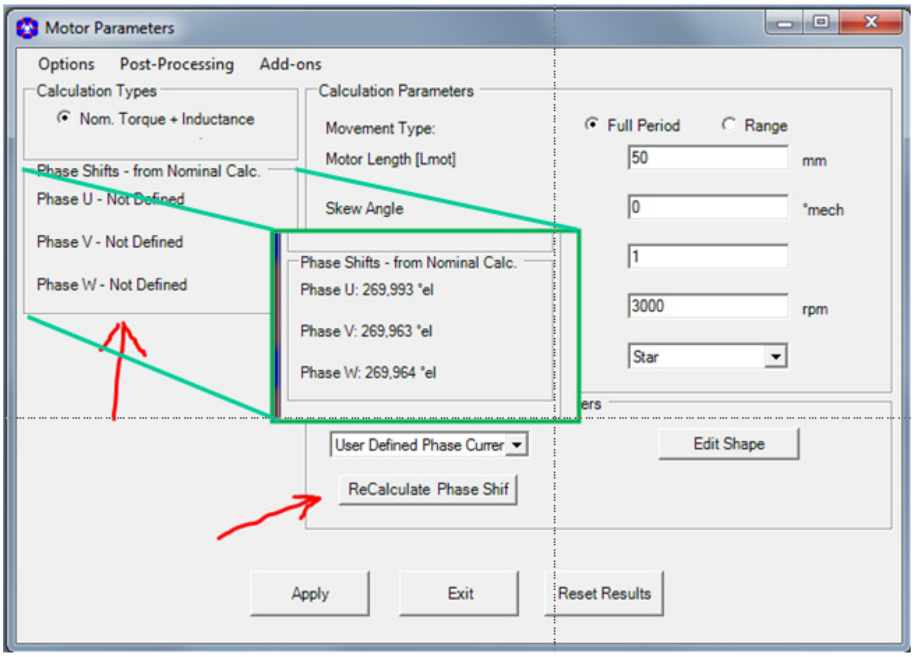


Fig. 212: SR motor - Determination of phase positions

The simulation results are then displayed in the same way as described in the previous chapters.

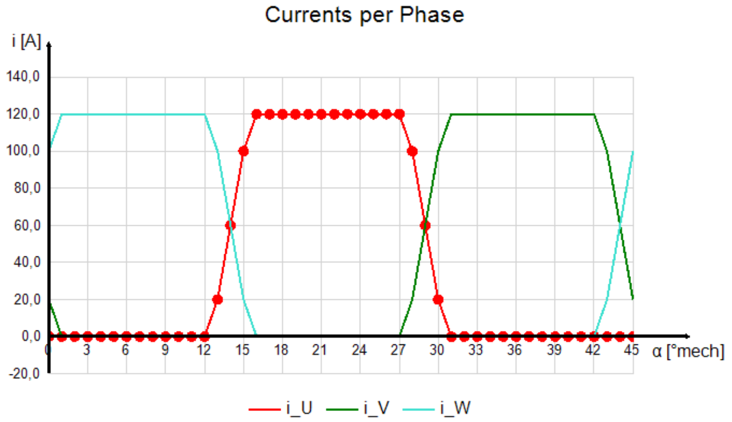
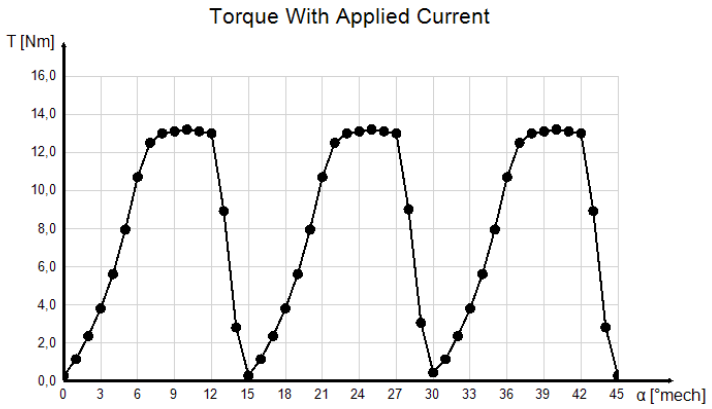


Fig. 213: SR motor - Results



6.5 Synchronous Reluctance Motor

The creation of a SR motor simulation model is initialized via *"File - New - Select Motor Type"*. At least one SR rotor topology (* .top) must be stored in the topology directory in order to display the *„SR - Motor“* menu item.

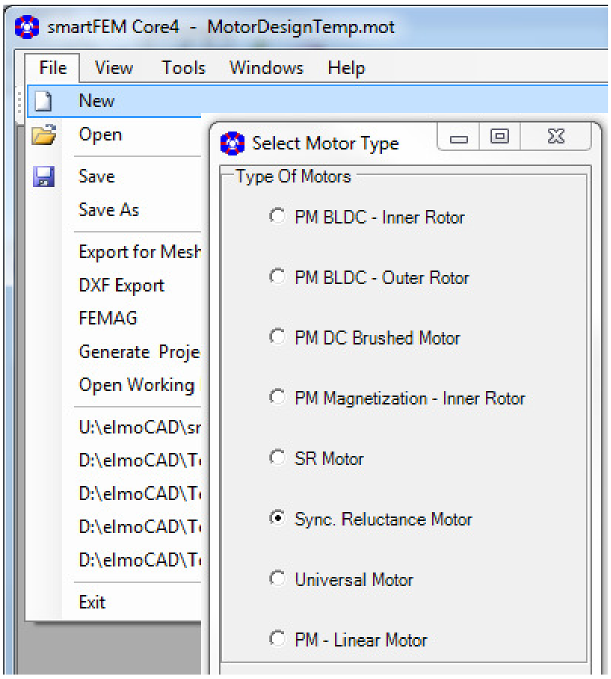


Fig. 214: Synchronous Reluctance Motor -Initialization

A motor model with 4 poles and 4 flow barriers (rotor topology CR_31_Synchronous_Relectance) with rotor position in d-axis is displayed.

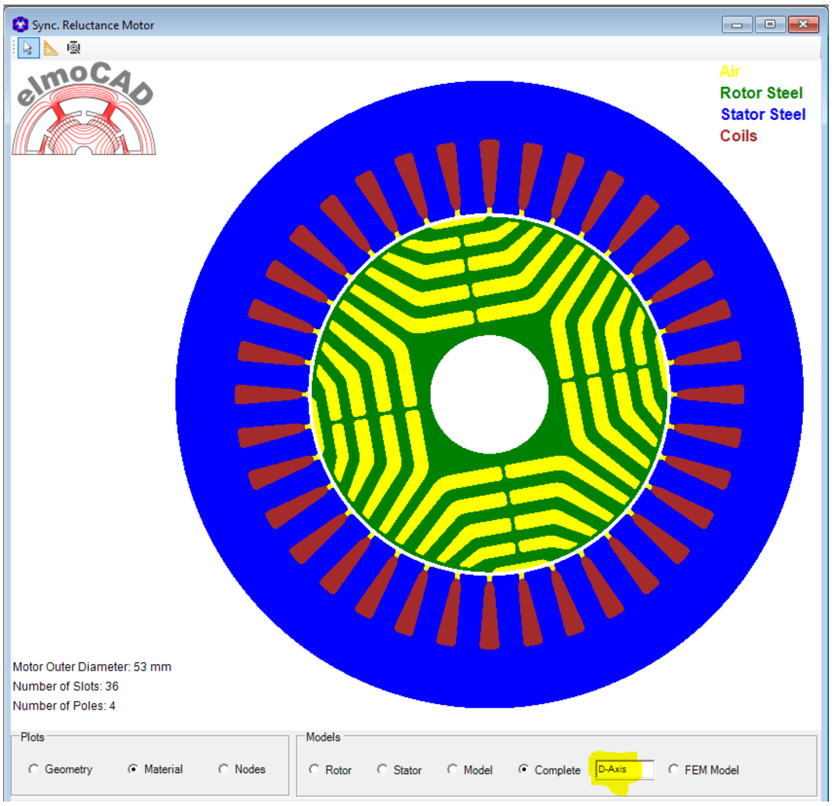


Fig. 215: Synchronous Reluctance Motor - 4 poles

Example: Results *„Torque vs. Theta“* from parametric Ld/Lq calculations inlc. phasor diagram of the maximum of torque.

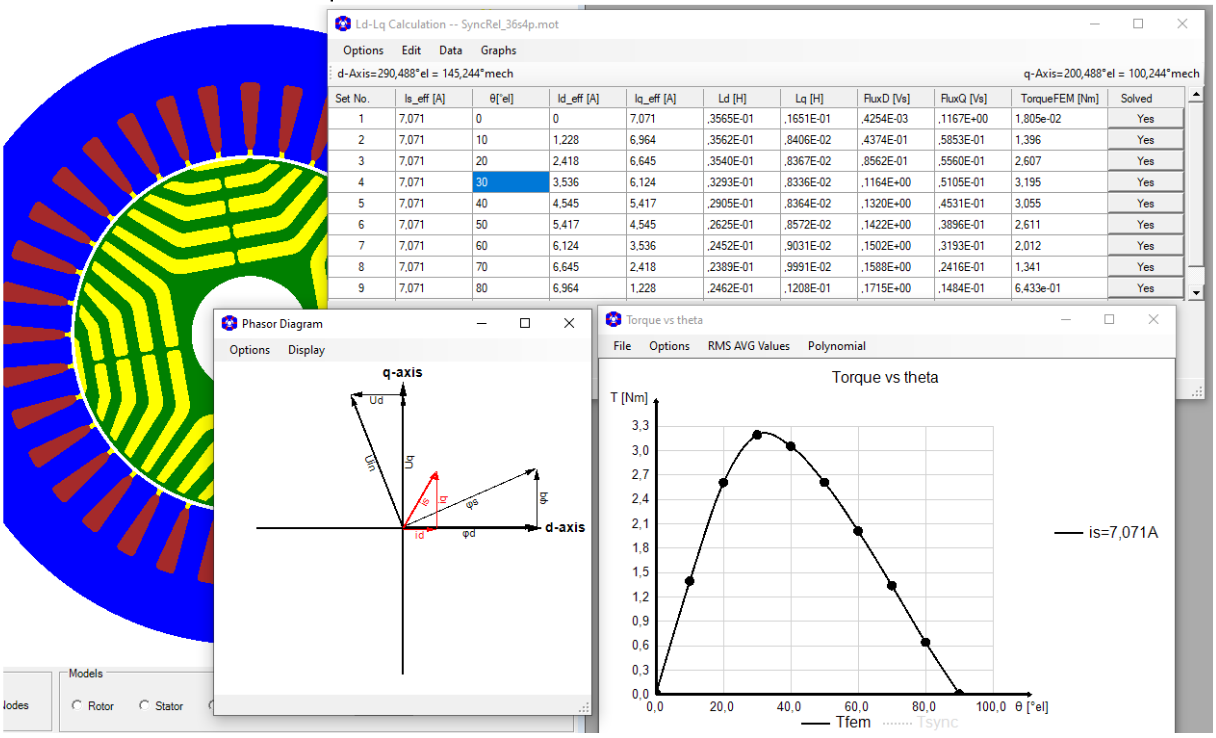


Fig. 216: Synchronous Reluctance Motor - Example of simulation results

6.5.1 Synchronous Reluctance Motor PM-Assisted

This type of motor is defined in smartFEM as *„PM BLDC - Inner Rotor“*. The name of the rotor topology is *„CR31_Synchronous_Reluctance_PM_Assisted“*.

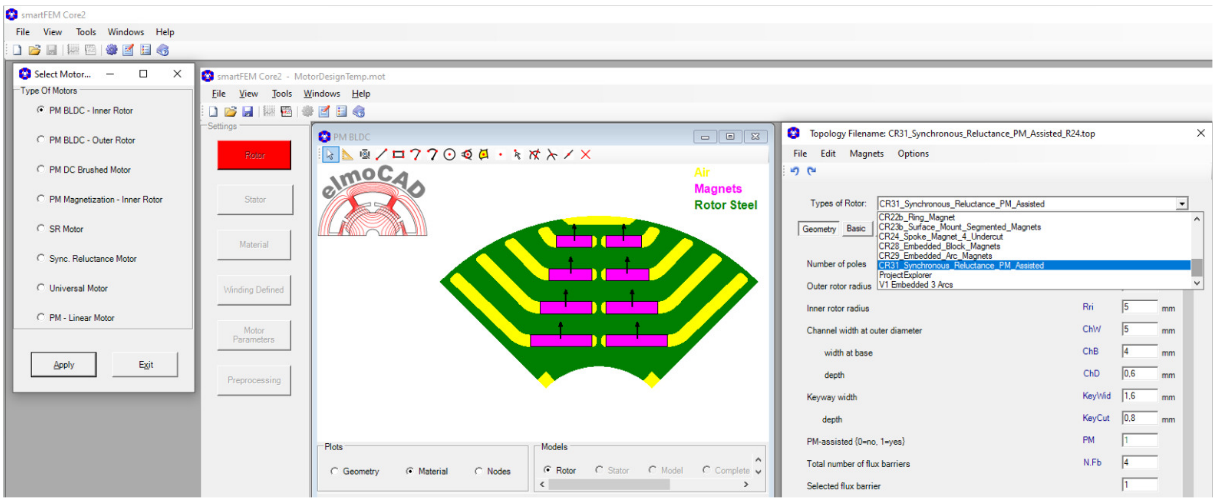


Fig. 217: Synchronous Reluctance PM-Assisted Motor - Rotor Topology

Example: Results of *“Torque vs. Theta”* from parametric Ld/Lq caculations incl. phasor diagram of the maximum of the torque.

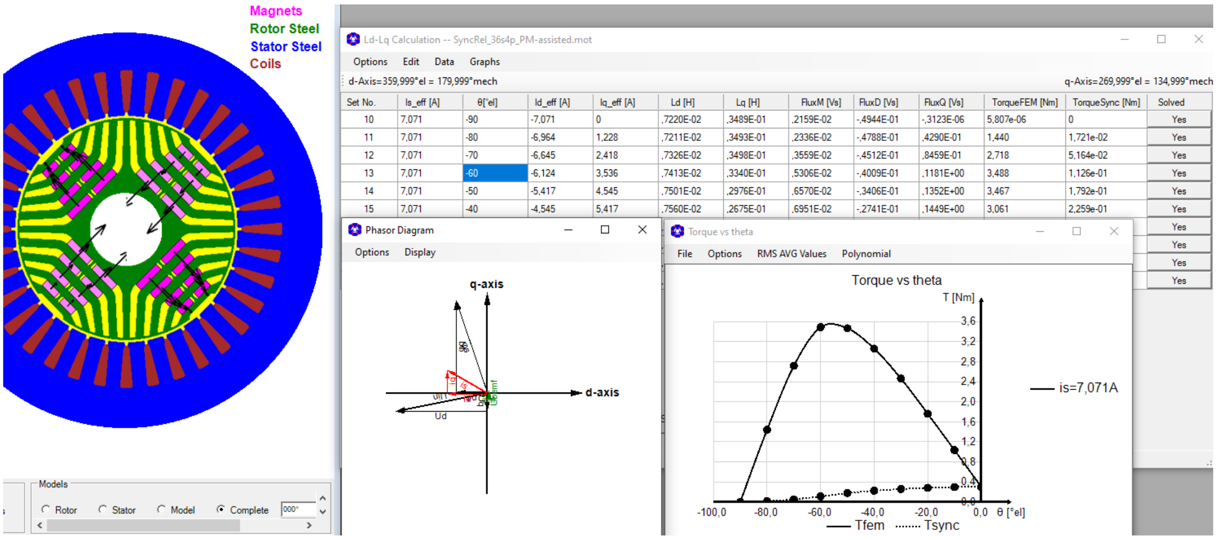


Fig. 218: Synchronous Reluctance Motor PM-Assisted - Example of Simulation Results

6.6 Universal motor

The creation of a Universal Motor model simulation is initialized via *"File - New - Select Motor Type"*. At least one universal rotor topology (* .top) must be stored in the topology directory, so that the menu item *„PM - Universal Motor"* is displayed.

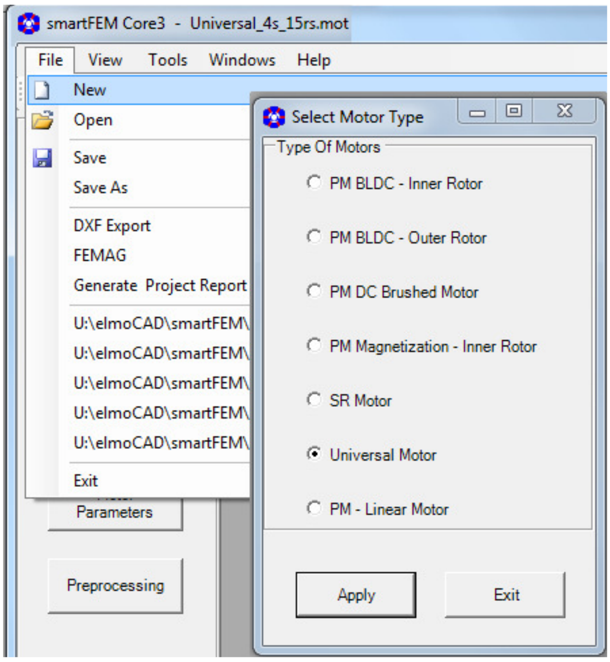


Fig. 219: Universal motor initialization

The modeling of a universal motor does not differ from the modeling of the other motor types apart from the rotor and stator separated windings and the setting in *„Motor Parameters"*.

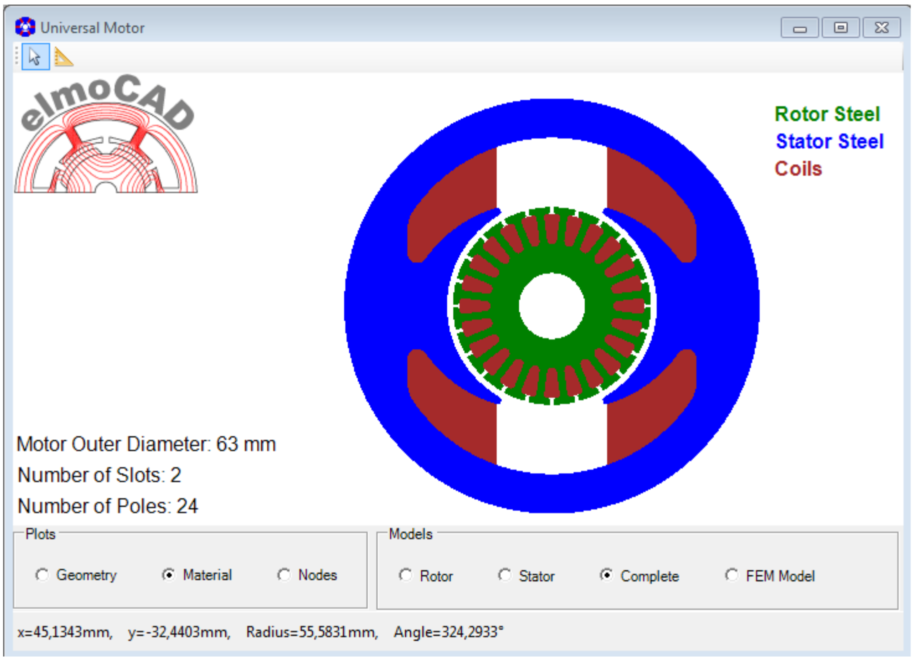


Fig. 220: 2-pole universal motor with 24 rotor slots

After "Preprocessing", the following calculation can be performed:

- Creation of plots of induction with/without rotor (or armature) and stator currents for the design of the magnetic circuit

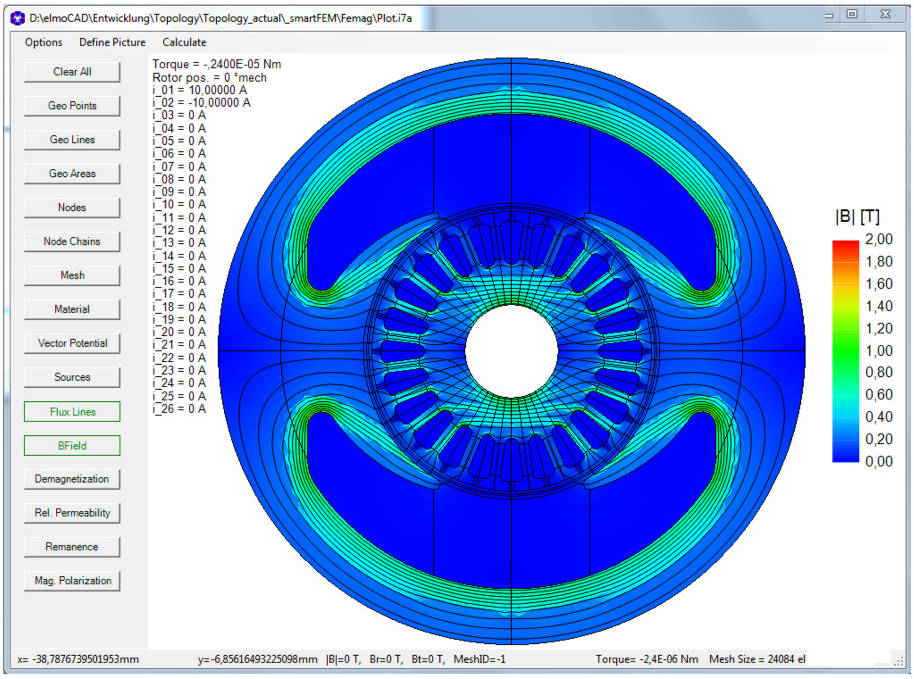


Fig. 221: Universal motor - Plot of induction and field lines

DC and AC motors can be simulated via the settings in *"Coil Examination - Current Parameters".* The simulation is performed in several steps:

- Calculation of the commutation sequence ("Commutation Sequence")
- Calculation of torque during commutation („Switching Torque“) Determination of the torque
- Calculation of torque in AC or DC operation ("Motor Torque")
- Calculation of the rotor position for the positioning of the brushes

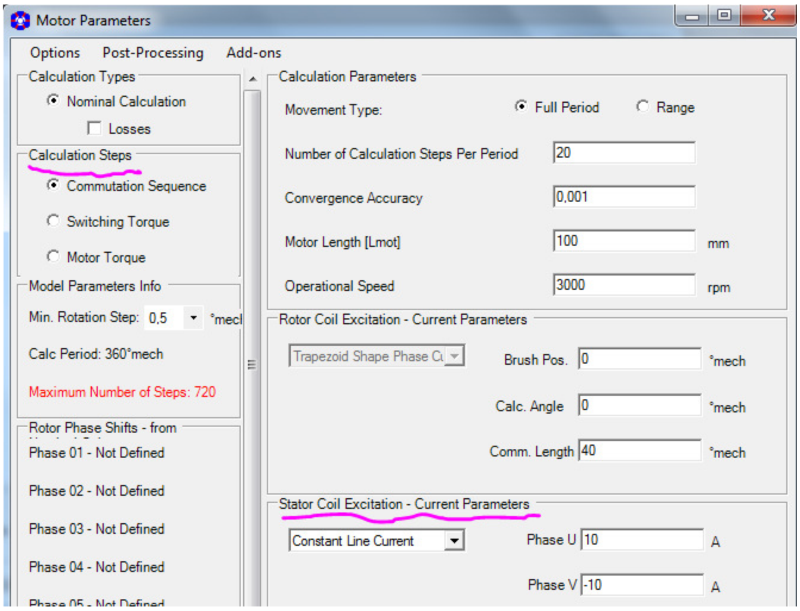


Fig. 222: Universal motor - Motor parameters

6.6.1 Calculation of "Commutation Sequence"

Since because of the missing magnets, *"Cogging Torque + BEMF"* cannot be determined, the *"Nominal Calculation"* is only possible without/with loss calculation. First, the basic parameters are set in *"Motor Parameters*" and the first simulation step *"Commutation Sequence"* is selected. The following calculations are performed after clicking *"Apply"* and *"Calculation"* buttons:

- Excitation of the field winding with DC current (amplitude) at high armature speed.
- Simulation without armature currents to determine the flux per winding depending on the winding scheme.
- Simulation with rotation of the armature.
- Determination of the induced voltage („*Transformer Voltage*") in the armature windings and their phase positions.
- Definition of the commutation sequence on the basis of the induced voltages.
- Determination of a first brush position based on the induced voltage or flux in the windings
- for a given armature position (windings with maximum flux or minimum induced voltage, respectively, begin to commute).

After performing this simulation step, the phase positions of all windings are shown in *"Motor Parameters"*. Coil no. 13 in slot 01 on the right and slot 13 on the left have the phase position 0 ° el.

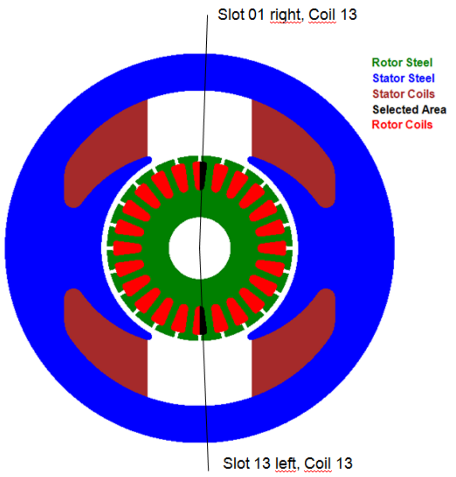


Fig. 223: Universal motor - Position of the winding 13

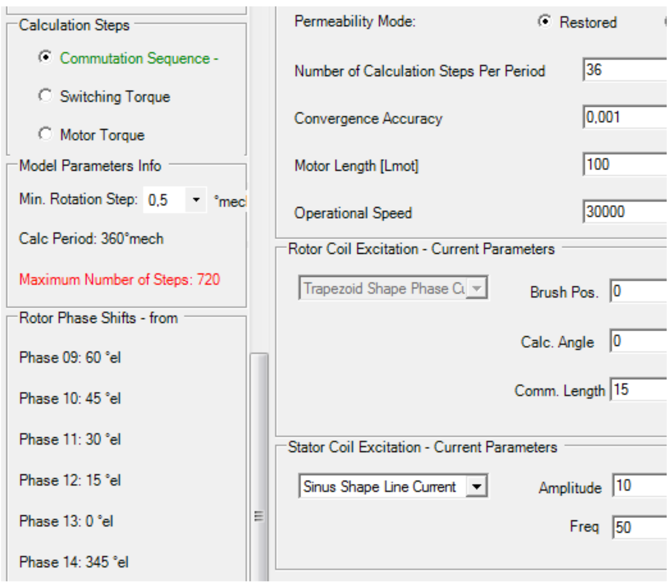


Fig. 224: Phasing of the coils

In *"Results"*, the following graph, among other things, is dosplayed:

"Transformer Voltage per phase": the graph for slot no. 13 (phase position 0 ° el with the individual calculation points) is highlighted in green.

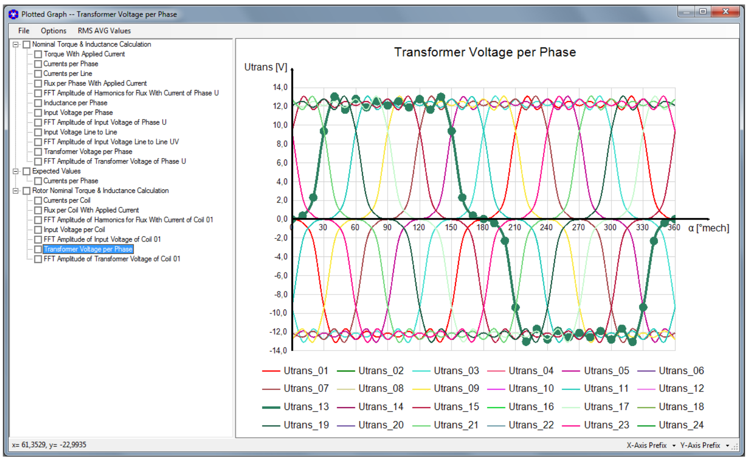


Fig. 225: Universal motor - Transformer Voltage per rotor phase (Rotor Current = 0A)

6.6.2 Calculation of torque during commutation "Switching Torque"

The settings are made in *"Motor Parameters - Calculation Steps*". Among others can following graphs be displyed in *"Results"*:

For the chosen example with 2 stator slots and 24 rotor windings are the calculations performed across the rotor positions 0 - 15°mech.

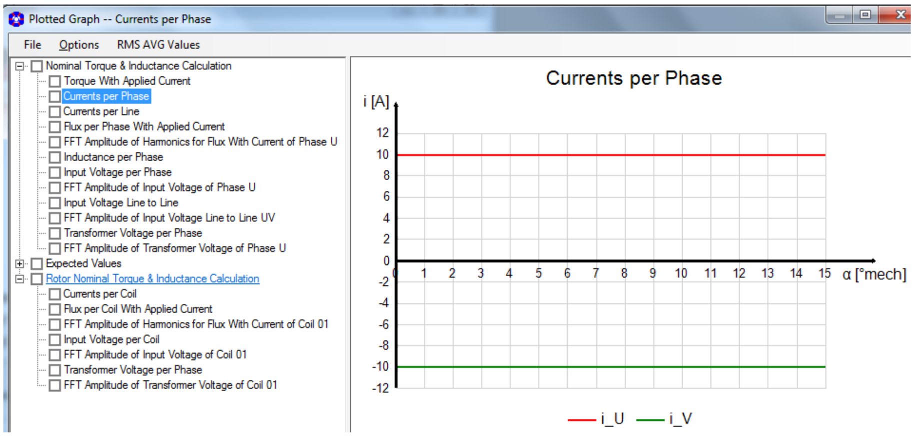


Fig. 226: Switching Torque - Stator currents

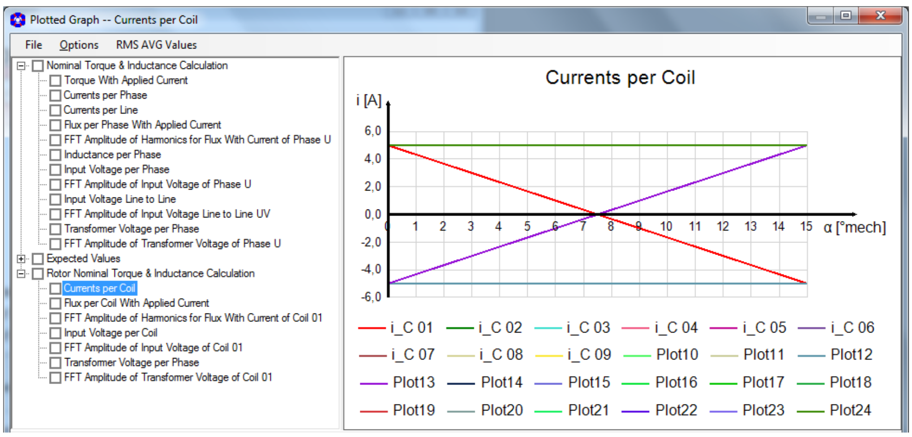


Fig. 227: Switching torque - Rotor currents

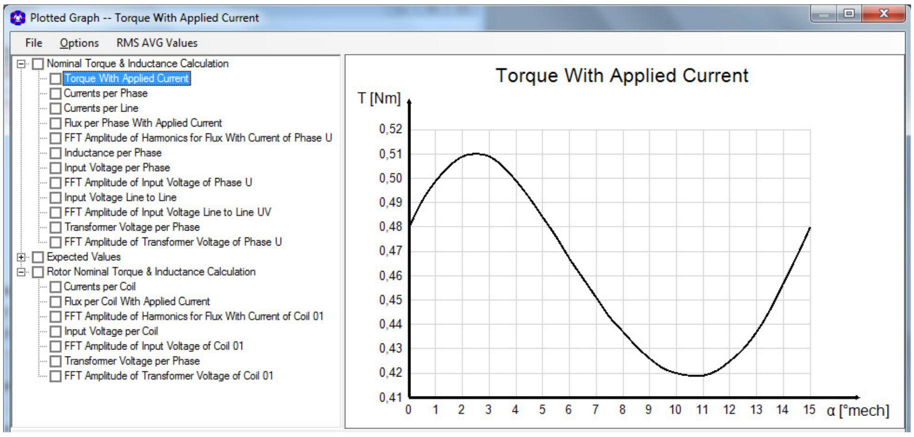


Fig. 228: Switching Torque - Torque

6.6.3 Calculation of torque in AC or DC operation "Motor Torque"

For DC operation, constant current must be selected in *"Motor Parameters - Coils Excitation".* For the results shown below, sinusoidal current was set, i.e., the simulation was performed as of an AC motor.

The unit of the abscissa is given in the time domain - in the example, the period of the sinusoidal stator current is 20 ms corresponding to a frequency of 50 Hz.

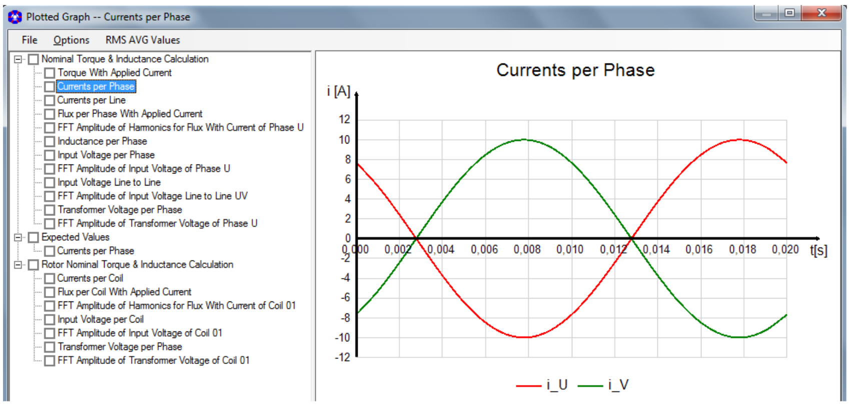


Fig. 229: Motor torque - Stator currents

The rotor currents from i_C 01 to i_C 13 have the same phase position. The rotor currents from i_C 14 to i_C 24 also have the same phase position but an offset by 180 ° el compared to the previously mentioned one.

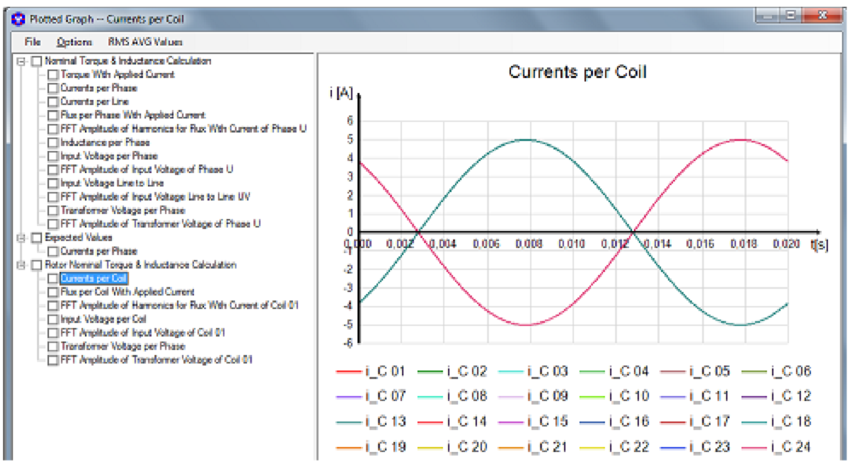


Fig. 230: Motor Torque - Rotor currents

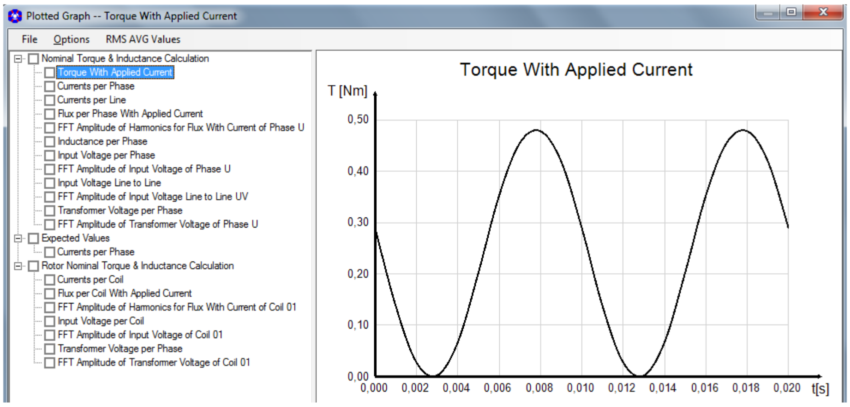


Fig. 231: Motor Torque - Torque of the AC motor

6.6.4 Calculation of the rotor position for the positioning of the brushes

For the calculation of the optimal position of the brushes, different brush positions, calculation angles and commutation lengths can be set via the menu sequence "*Motor Parameters - Coil Excitation - Current Parameters*". *"Motor Torque"* is selected in *"Calculation Steps"*. The results are displayed as usual in the diagrams in the *"Results"* menu.

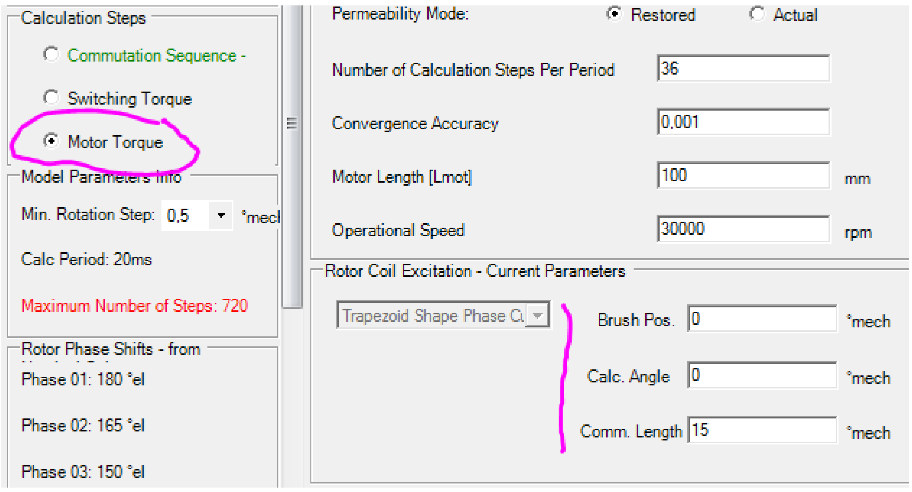


Fig. 232: Universal motor - Positioning the brushes

6.7 PM - Linear motor

The creation of a linear motor simulation model is initialized via "*File - New - Select Motor Type*". At least one linear motor topology (* .tol) must be stored in the topology directory so that the menu point „*PM - Linear Motor*“ can be displayed.

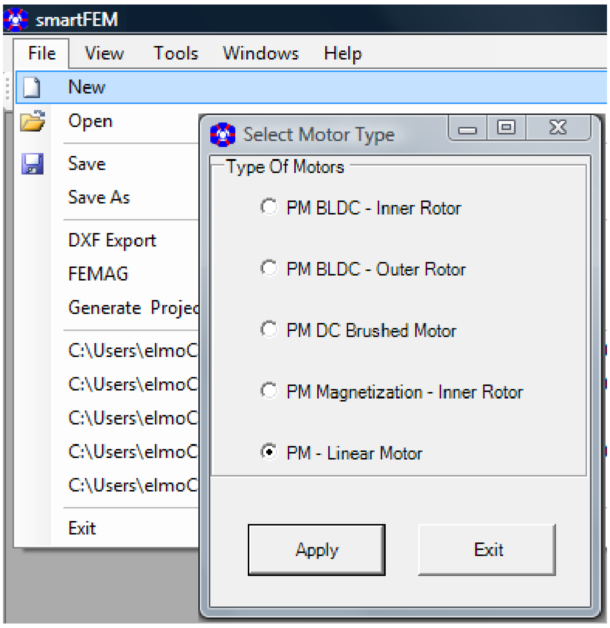


Fig. 233: PM-Linear motor -Initialization

6.7.1 Modellbildung

The modeling of a linear motor differs from the modeling of the other motor types only that rotor geometry and stator geometry are parameterized in a *"Geometry"* topology. All other functions are the same - except for the calculation of the iron and magnetic losses and motor diagrams.

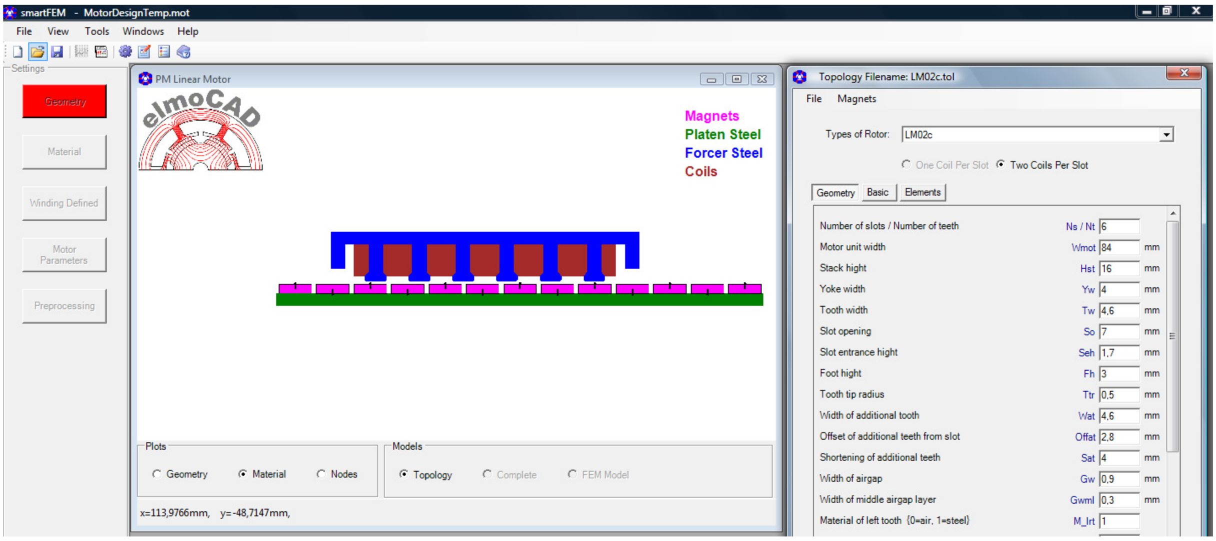


Fig. 234: PM -Linear motor - Example of a geometry model

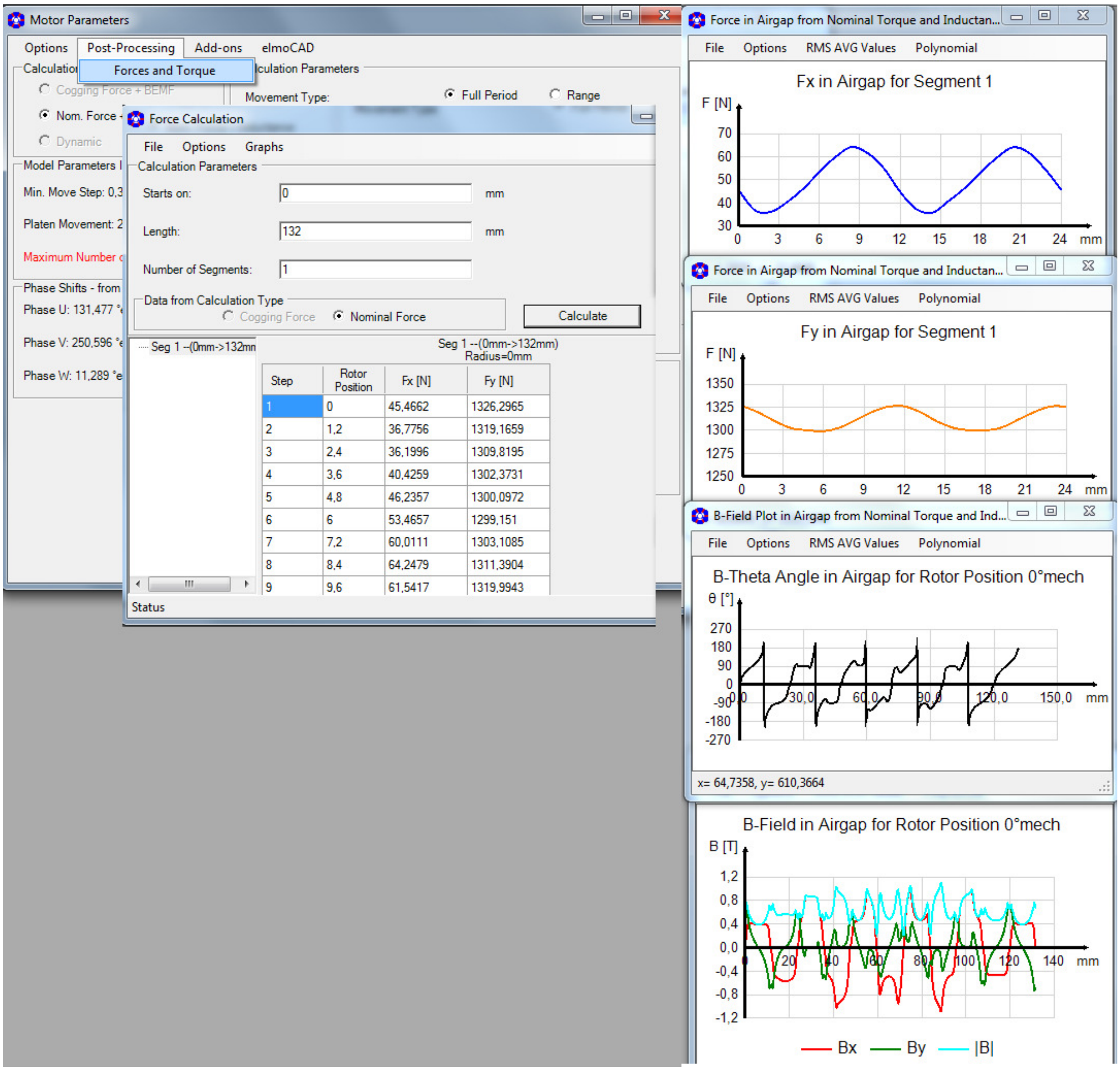


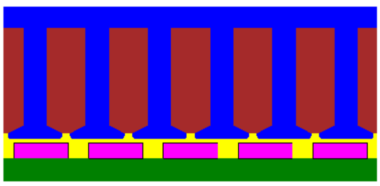
Fig. 235: PM-Linear motor - Representation of forces and flux density in the air gap

6.7.2 Calculation of Losses

Losses of linear motors are only calculated forperiodic models because for this only the Fourier transformations can be executed.

Following topologies are periodic respective can be switched between periodic and non-periodic.

- ***„H3LM2 Linear motor - periodic“*** ***„H3LM3 Linear motor - full slot winding - periodic“***



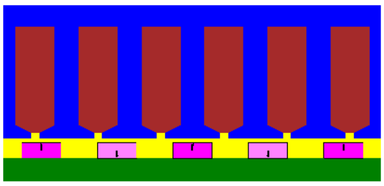


Fig. 236: PM - Linear Motor - periodic models H3LM2 and H3LM3

- ***„LM021c“***

The Paramter *Type of Model"* was added to paraeter group *„Geometry"*. With this can the model be switched between non-periodec and periodic. When the periodic model selected then all non-periodic elements of the geometry are omitted.

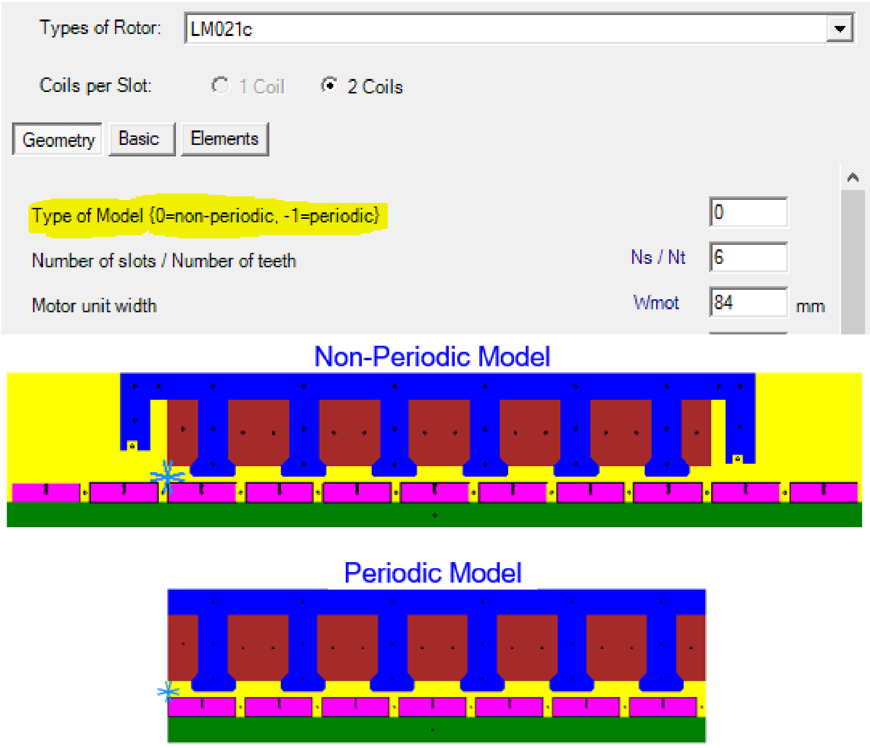


Fig. 237: PM - Linear Motor - LM021c, switching periodic / non-periodic

- ***"LMxxx"***

All other liniear motor topologies will be adapted accordingly.

The loss calculation is carried out like rotational machines by selecting ***"Losses"*** in ***"Motor Parameters"***. After calculation ***"Nominal Force + Inductance"*** is performed are the losses determined in ***"Postprocessing"***.

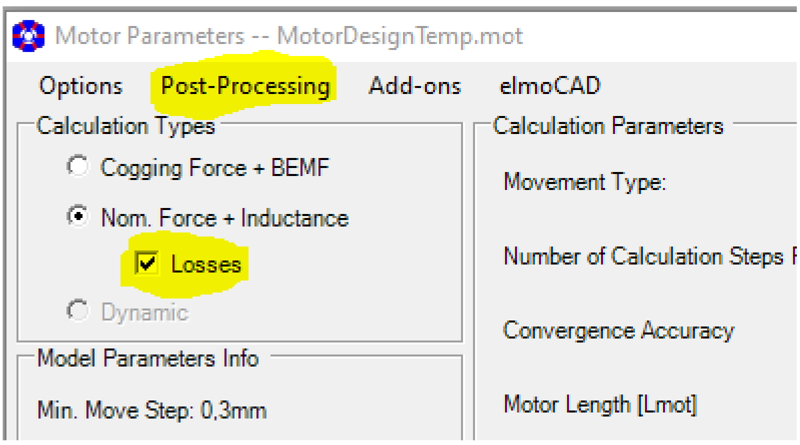


Fig. 238: PM - Linear Motor - Loss calculation

6.8 Actuator

Geometry models of actuators such as valves can be designed with CAD software according to DXF import rules of smartFEM and be simulated after DXF import.

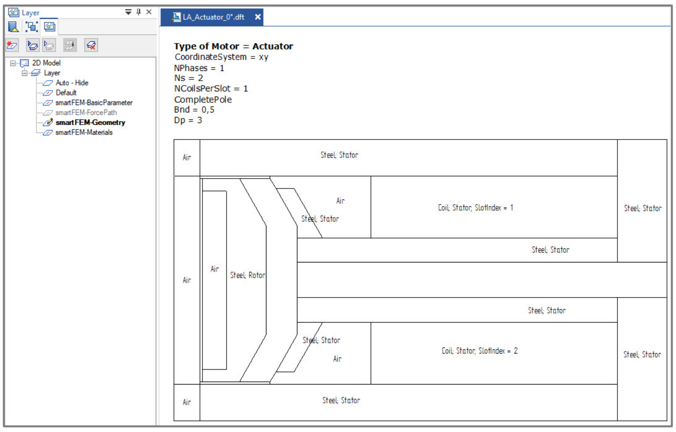


Fig. 239: Actuator - CAD drawing

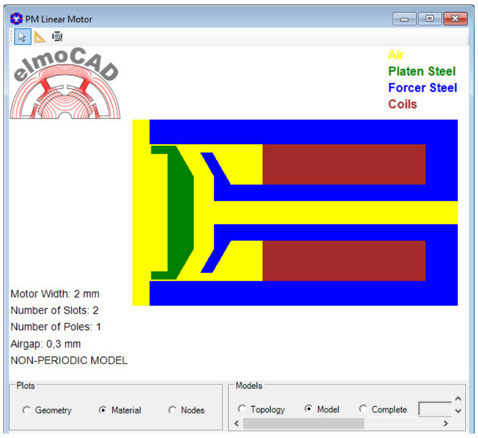


Fig. 240: Actuator - smartFEM model after DXF import

The CAD drawing must contain following basic parameter as text:

- ***„Type of Motor = Actuator“***
- ***„CompletePole = True/False“*** oder ***“CompletePole“*** bzw. ***„HalfPole“***

Further parameter can be added optionally:

- „CoordinateSystem = xy“ (defaultt value, if not specified)
  or „... = rz“ for rotation-symmetric models
- Additional parameter accordingly other machine models such as:
  NPhases, Ns, NCoilsPerSlot, Bnd, Dp, etc.

Actuators are actually treated as liniear motor will be later available as own machine type. The topology name for DXF import of actuator drawings is ***„LA_CADdata“***.

To determine the forces which are applied to the moving part (named as *„platen“*) the ***„ForcePath“*** must be defined in the drawing as polygonal line and be marked by text *„ForcePath“.* In case that the forces should be calculated at different positions of the platen must the drawing also contain as line as vector which is marked by the text ***„MoveVector“***. The movement of the platen is carried out from start point of the vector in direction of the end point. The maximum possible path correspondens to the length of the vector.

To define the origin P(0,0) a point has to be drawn at an related position, which is located i.e. on the middle line with y=0 [P(0,0) = P(x,0)].

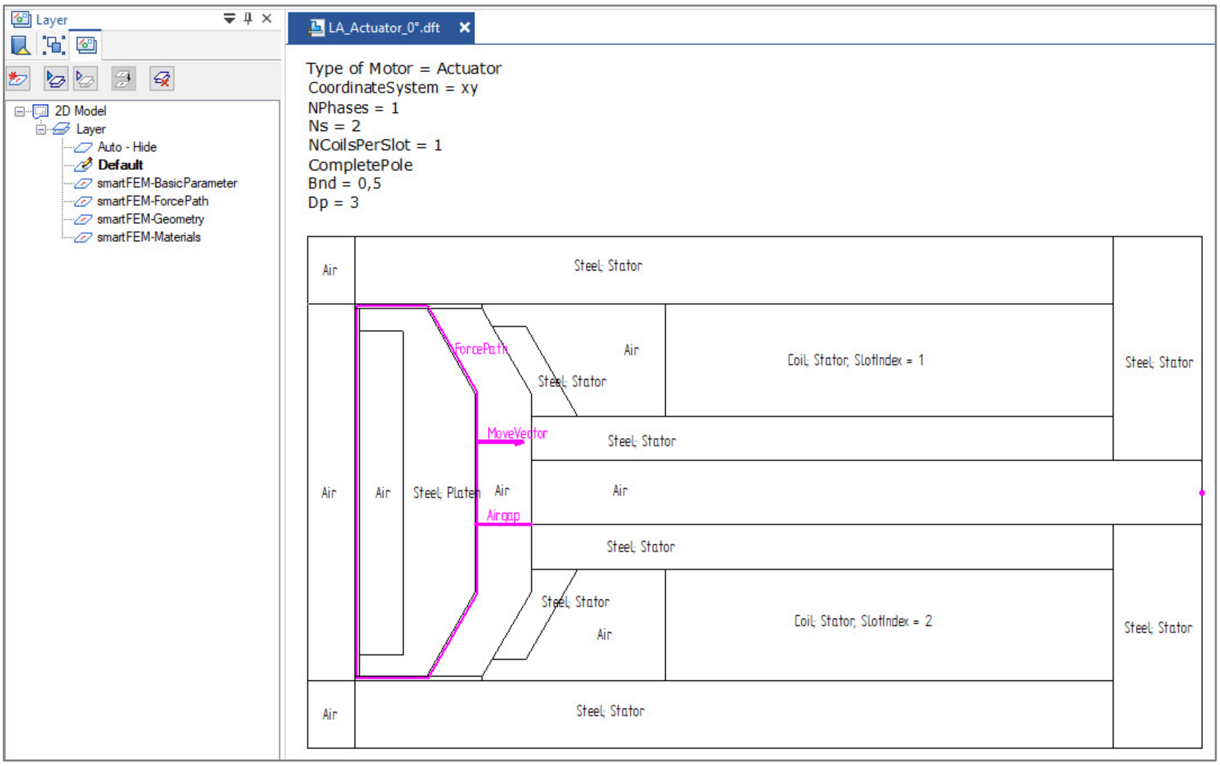
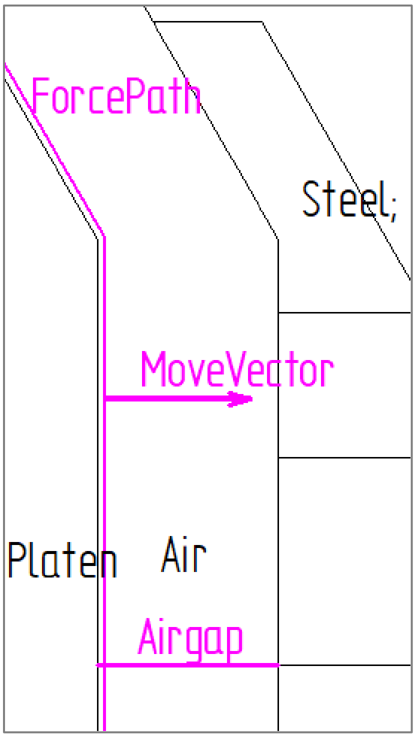


Fig. 241: Actuator - ForcePath and MoveVector



The polygonal line „*ForcePath*“ must deflect anti-clockwise arround the platen completely in air.

The „*MoveVector*“ can be placed at any position. All elements of the geometry which are inside the area of the „*ForcePath*“ are displaced to a the related new position.

The airgap is defined by the user and can be drawn as a line indicated by the text *"Airgap"* with between platen and stator. The length will be displayed after DXF import as parameter and will also be indicated in the graphic window.

Fig. 242: Actuator - ForcePath and MoveVector

The geometry of the moveable part of the geometry must be surrounded by air and be connect by a line to the fixed part of the geometry.

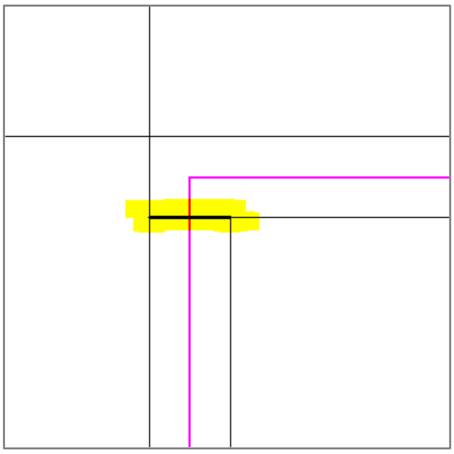


Fig. 243: Actuator - Connection of the moving part with the fixed part of the geome

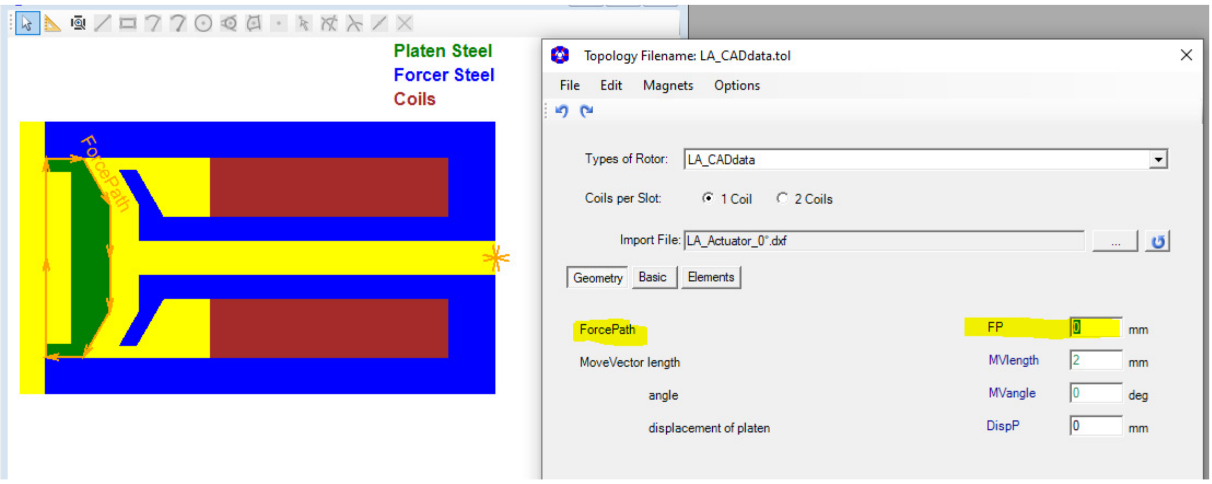


Fig. 244: Actuator - presentation of the "ForcePath" in smartFEM

Fig. 245: Actuator - presentation of the "MoveVector" and displacement of the platen

Results after calculation of „*Nominal Torque*“ and „*Plots*“:

- Force in dirction of the „*MoveVector*“ vs. „*Displacement*“
- Field plots

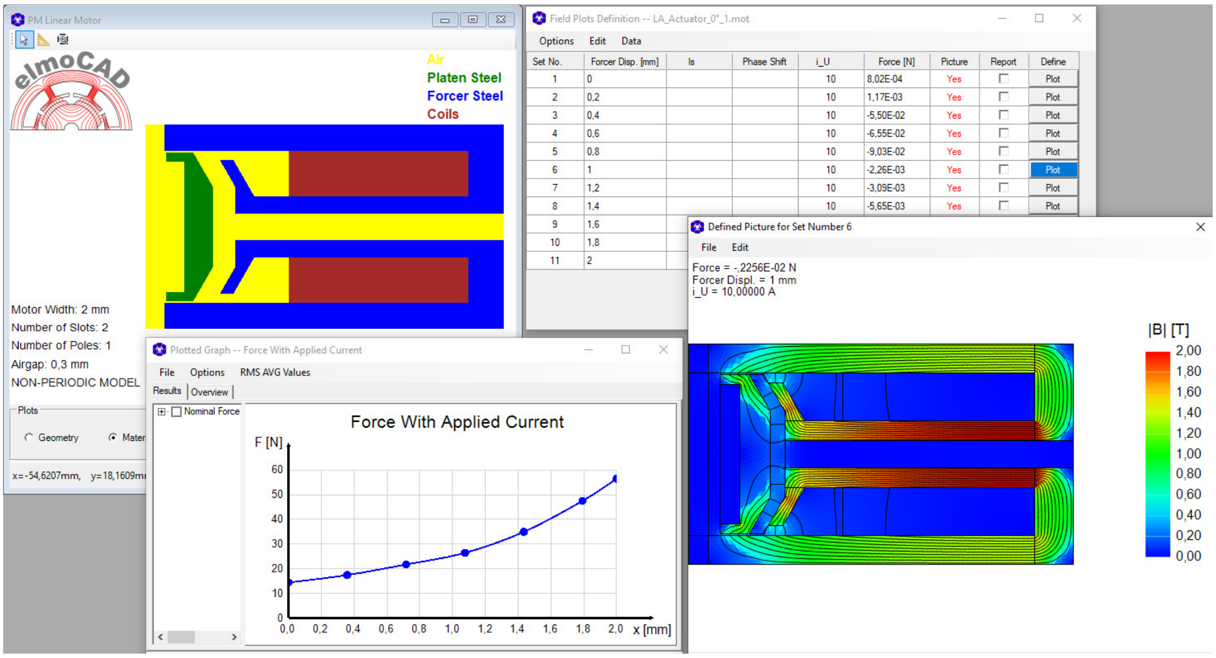
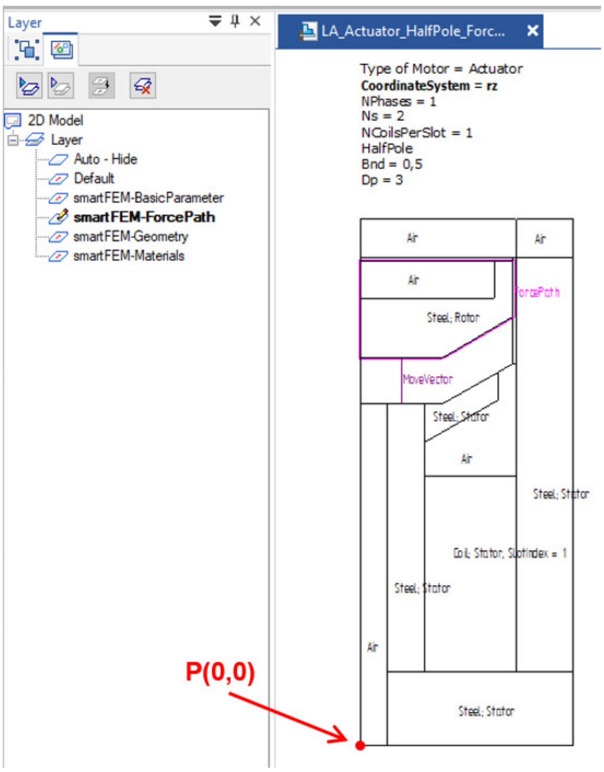


Fig. 246: Actuator - Simulation and Results

For rotation-symmetric actuators only the half of the geometry is allowed to be drawn beginning from the center line respectively symmetry axis to the right side. The origin P(0,0) should be drawn at the lower left corner. It is also mandatory to specify the basic parameter ***„CoordinateSystem = rz“***.

This functionality is still not implemented!



7 Interfaces

7.1 Export of result data as tabled text

All result data can be copied into other software tools by „Copy/Paste“ and partly also can be saved directly in tabled text files.

Examples:

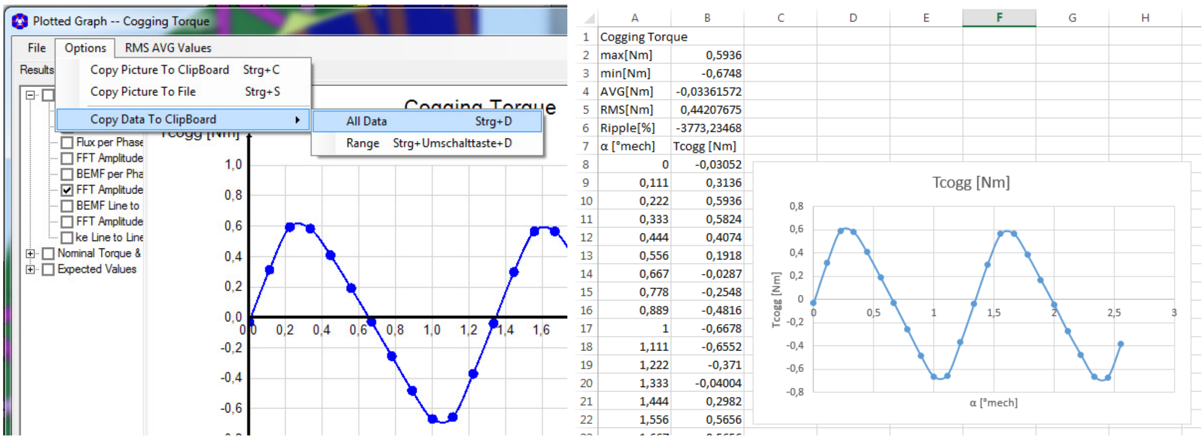
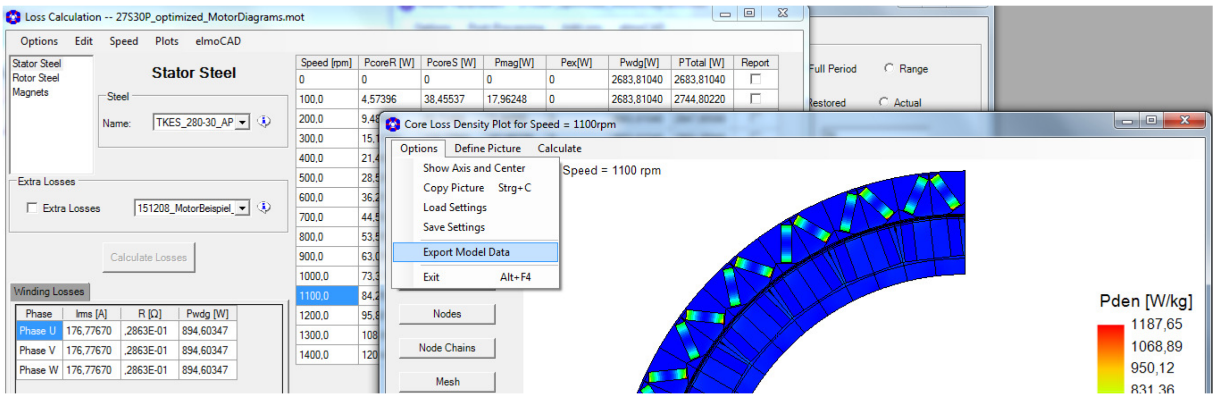


Fig. 247: Export of result data as text - Cogging Torque



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
26792																
26793																
26794	MESH PROPERTIES															
26795	Nr.	Material	Area[mm2]	CenterR[mm	CenterAngle	Br[T]	Bt[T]	|B|[T]	|H| [kA/m]	-Hc [kA/m]	μr	i_den [A/mn	Mr [mT]	|J| [T]	Pdens[W/kg]	Nodes
27276	481	ryok	0,47527113	191,3968	91,9393374	0,08971814	-0,89424002	0,89872941	0,11416928	0	6264,25996	0	0	0,89858595	0,923151578	(2789 2448 2449)
27277	482	ryok	0,47525208	192,0001	91,9521218	0,088072	-0,87380929	0,8782365	0,11067081	0	6314,92924	0	0	0,87809743	0,893510713	(2788 2448 2789)
27278	483	ryok	0,47520556	190,5298	91,7629014	0,0778048	-0,90757444	0,91090337	0,11627665	0	6234,04412	0	0	0,91075725	0,964858027	(2790 2449 2450)
27279	484	ryok	0,47522462	191,133	91,7762931	0,07600725	-0,88613072	0,88938448	0,11256806	0	6287,30444	0	0	0,88924302	0,940062934	(2789 2449 2790)
27280	485	ryok	0,47520534	189,6647	91,5848459	0,06593247	-0,92340466	0,92575551	0,11887695	0	6197,10299	0	0	0,92560612	1,000113297	(2791 2450 2451)
27281	486	ryok	0,47520534	190,2677	91,5988708	0,0638589	-0,89942231	0,90168645	0,11468344	0	6256,69438	0	0	0,90154234	0,992668512	(2790 2450 2791)
27282	487	ryok	0,47527138	188,8013	91,4051528	0,0540221	-0,94320615	0,94475194	0,12233342	0	6145,57786	0	0	0,94459821	1,002184352	(2792 2451 2452)

Fig. 248: Export of result data as text - Model Data

7.2 Documentation "Project Report"

smartFEM offers the possibility to create a project report by menu *„File"→"Generate Project Report*", which is saved in a RTF-formetted textfile and can be edited by popular text programs. The contents of the report (design data, calculation results, and diagrams) can be selected by the user.

The diagrams can be selected at the related simulation results:

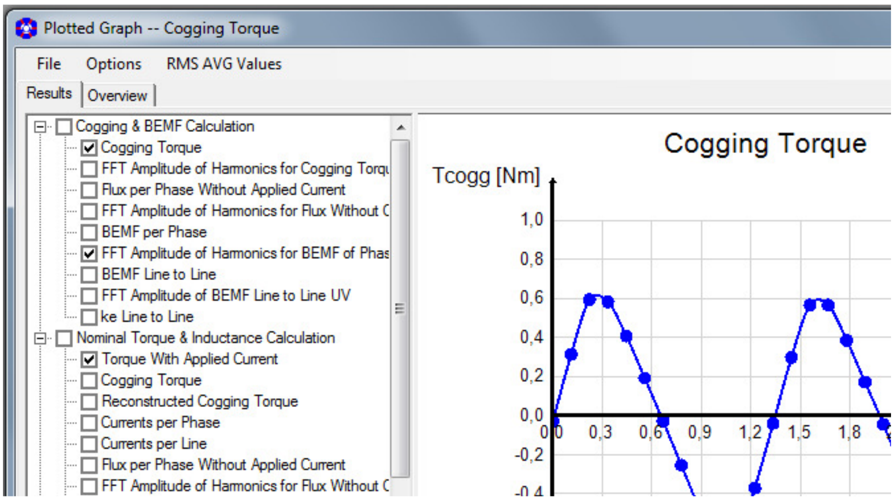


Fig. 249: Project Report - Selection of diagrams

Diagrams in general and design parameter and calculation results as text can be selöected in a pop-up windows after selction of the menu *"Generate Project Report":*

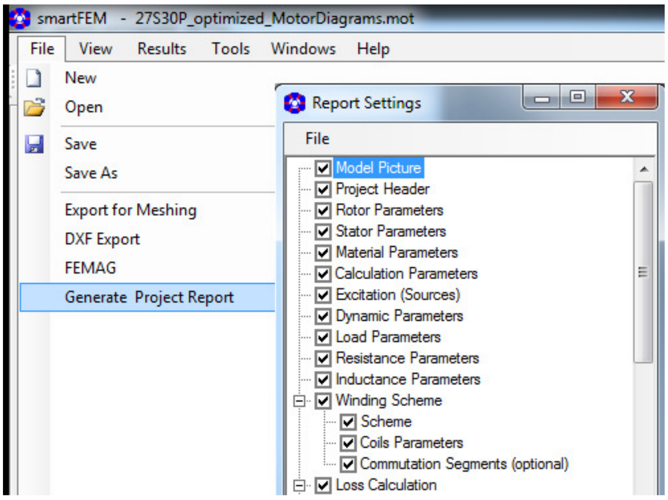
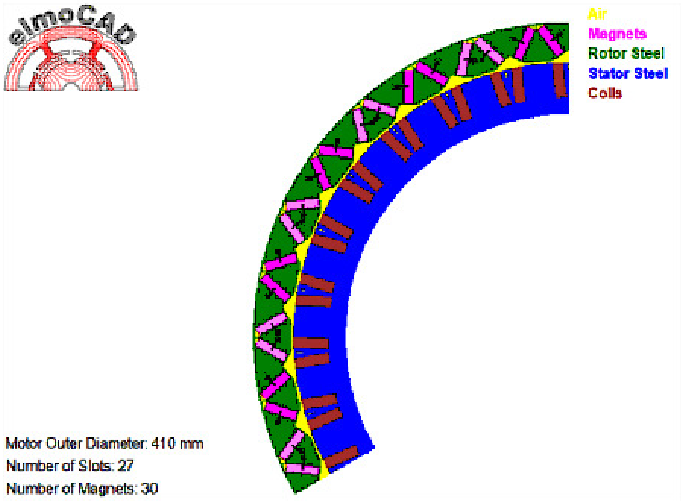


Fig. 250: Project Report - selection of design parameter and results as text

Example:

MOTOR MODEL: 27S30P_optimized_MotorDiagrams.mot



PROJECT HEADER	
Project Number:	2.9.00
Project Name:	smartFEM User Guide
Customer:	
Date of design:	15.02.2017 09:49:52
Designer:	ELMOCADLT01A\gh
Comment:	
smartFEM Version:	Release 2.9.00
FEMAG Version:	7.9.34.07

MOTOR TYPE PMBLDC

COGGING TORQUE + BEMF CALCULATION

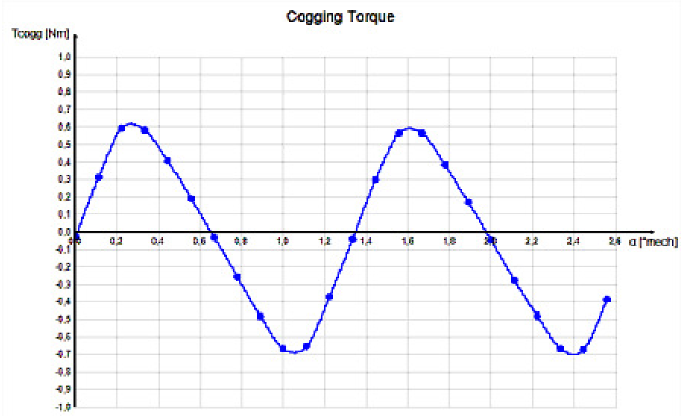


Fig. 251: Projekt Report opened by Microsoft® Word

7.3 FEMAG

smartFEM uses FEMAG for the numerical calculation of all the results, which are determined on the basis of the finite element method (FEM). For this, smartFEM transfers all necessary information to FEMAG and controls all FEMAG processes

- to create the simulation model required for analyzing and evaluating the numerical computations - triggered by pressing the *"Preprocessing"* button.

- to perform all analyzes and evaluations - triggered by pressing the *"Calculation"* button.

smartFEM receives all relevant data from the FEMAG results files for further calculations, evaluations and graphical representations.

Running FEMAG processes can be contolled and aborted at any time by pressing the *"Calculation Running"* button without losing any data.

The user can start FEMAG at any time directly by pressing the *"FEMAG"* button. It will be used in the version that was previously set through *"Tools → smartFEM Settings → User FEMAG Exe File"*. By clicking on the *„FEMAG"* button with the right mouse button, a following selection menu is offered:

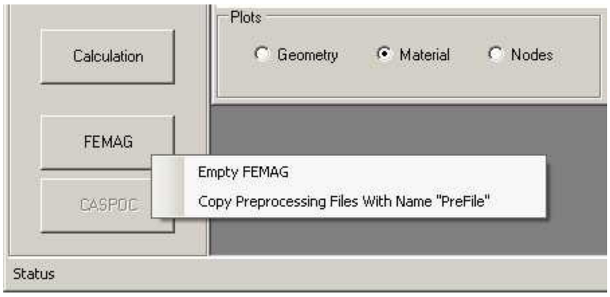


Fig. 252: Starting FEMAG from smartFEM

- *"Empty FEMAG"*
  FEMAG is started. The user can create a new FEMAG model or edit an already existing one.

- *"Copy Preprocessing Files With Name" PreFile ""*
  The current smartFEM model is available as a copy in the specified FEMAG directory under the names *"Prefile.a7"* and *"PreFile.i7"*. At the same time, FEMAG is started and the user can load and edit the model using the FEMAG command *"Select Input File"*. Changes and evaluations to the model made by the user are not accepted by smartFEM.

7.4 CAD - DXF Import

7.4.1 Topologies

For the import of 2D rotor and stator geometries created with CAD systems, special topologies have been implemented which allow the import of geometry data in the DXF or CSV format:

- „PM BLDC - Inner Rotor"
  - ***CR_CADdata.top***
  - ***CS_CADdata.top***

- „PM – BLDC Outer Rotor", „PM DC - Brushed Motor"
  - ***OCR_CADdata.top***
  - ***OCS_CADdata.top***

- „Synchonous Motor"
  - ***Sync_CADdata.top***

- „SR Motor"
  - ***SR_CADdata.top***

- „Sync. Reluctance Motor"
  - ***SyncRel_CADdata.top***

- „Universal Motor"
  - ***UR_CADdata.top***

- „Linear Motor"
  - ***LM_CADdata.top***

  "Actuator"
  - ***LA_CADdata.top***

First they have to be opened like the "usual" topologies, before an import file can be selected.

7.4.2 DXF Import - Keywords and Parameter

The basis for the implementation of the DXF interface is the documentation named *"Autodesk_2011_dxf_reference_v.u.25.1.01"* by Autodesk, Inc., USA.

The entities "*POINT*", *"LINE"*, "*LWPOLYLINE*", "ARC", *"CIRCLE", „TEXT"* und *„MTEXT"* containing smartFEM relevant data are selected from the DXF files and all the information required for the simulation is generated:

- POINT: The origin P(0) of the smartFEM coordinate system has the coordinates x = 0 and y = 0 and represents the center point of all rotor and stator geometries.

  Note:
  If the origin of the coordinates exported to a DFX file from a CAD system is not at the center of the rotor or stator geometry (= intersection point of the right and left sector line of the pole), an additional point in this point has to be created in the CAD.

- ARC,
  CIRCLE,

- LINE,
  SPLINE: Out of the the selected information, the start and end points are generated. For arcs and circles, the center points are generated and numbered form 1 to n, taking a „PickPointRadius“ into account.

- TEXT: The following texts are required as additional information in smartFEM:

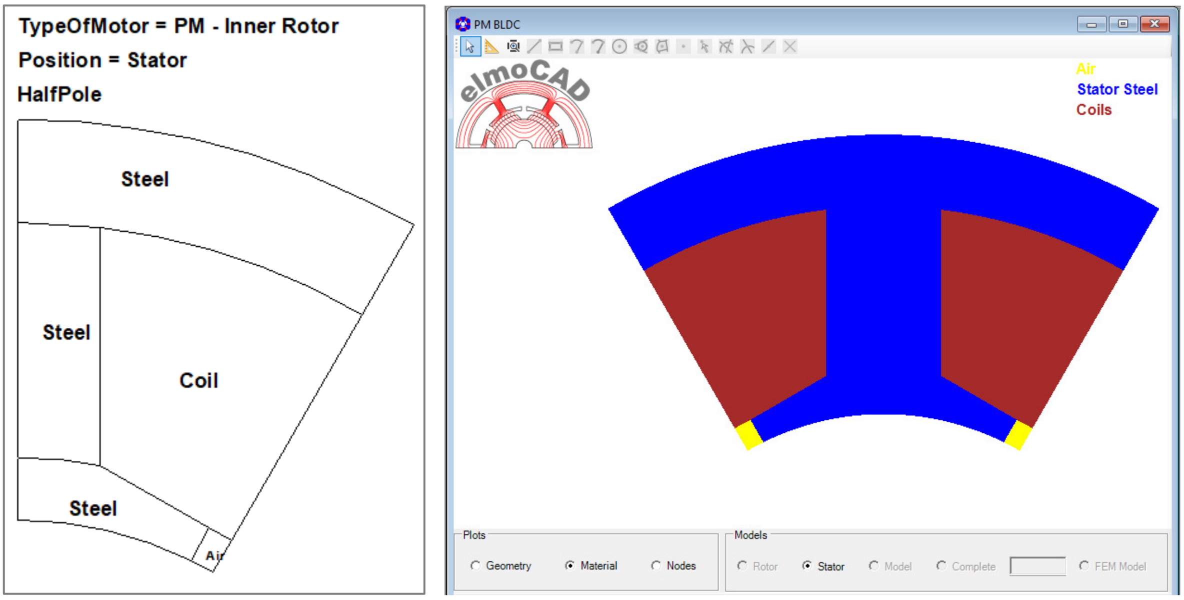


Fig. 253: DXF-Import - Example of the import of a stator geometry

Following keywords and parameter must be obgliatory or can be optionally defined in the CAD drawing:

<u>Obligatory Keywords and Parameter</u>

Keyword	Parameter	Description
Type of Motor	= PM - Inner rotor	Motor type: PM Inner rotor
	= PM - Outer rotor	PM outer rotor PM brush motor
	= SR - Motor- inner rotor	Switched Reluctance motor
	= Sync. Reluctance Motor - Inner Rotor	Synchronous Reluctance Motor
	= Universal motor	Universal motor
	= PM - Linear motor	PM_Linear motor
	= Actuator	Actuator

Position	= Rotor = Stator	Geometry type: Rotor Stator Note: When linear motors are concerned, the geometries below the air gap are defined as *"Rotor"*, and those above as *"Stator"*

Optional Keywords and Parameter

Keyword	**Parameter**	**Description**
Not used keywords have the value = False	[…] = default value if no parameter is specified	
HalfPole	[= True]	The geometry describes a half pole and is mirrored in the FEM model.
CompletePole	[= True]	The geometry describes an entire pole. It is not mirrored in the FEM model.
	[= [True;] Asymmetric]	The geometry describes a whole geometrically unsymmetrical pole.
MirrorPole	[= True]	The pole geometry is mirrored in the FEM model.
	= False	The pole geometry is not mirrored.
SegmentedGeometry	[= True]	A segemnted geometry with n poles per segment.
CompleteGeometry	[= True]	The complete geometry with all stator and rotor poles (360 °mech). It can be **non-symmetrical** with respect to the individual poles and thus **non-periodic** with respect to the BEMF. It is adopted without changes to the FEM model. The FEM calculations are performed with 360 °mech rotation of the rotor.
	[= True; Periodic]	The complete geometry is periodic with respect to the BEMF. The FEM calculations run over the BEMF period [° mech].
CoordinateSystem	[= xy]	The geometry is designed in xy section
	[= rz]	The geometry is rotaion-symmetric

The following texts is required to assign type of material to areas. The anchoring points of the text boxes must lie in the corresponding area elements. Many parameters have to be separated by the character semicolon ";". No specific order of parameters is prescribed.

The areas defined in this way can be selected in the geometry window of the topology with the mouse and be edited. Areas without an assigment of a material type are treated as air.

Keyword	Parameter	Description
Magnet		Material = Magnet
	[; MaterialNo = #] or [; MatNo = #]	Material number according to the material data defined in *"Materials"* for the model.
	[; Length = #.##]	Relative magnetic length in the axial z-direction [% of the motor length "*Lmot*"]
	[; MagAngle = #.##]	Angle of the magnetization direction with respect to the x-axis [°mech]
Steel		Material = electrical sheet
	[; MaterialNo = #] or [; MatNo = #]	Material number according to the material data defined in "*Materials*" for the model.
	[; Length = #.##]	Relative length in the axial z- direction [% of the motor length Lmot]
Coil		Material = coil
	[; MaterialNo = #] or [; MatNo = #]	Material number according to the material data defined in "Materials" for the model.
	[; SlotSideIndex = #]	Indicates on which side of the slot the coil is lying.
[Air]		Material = air

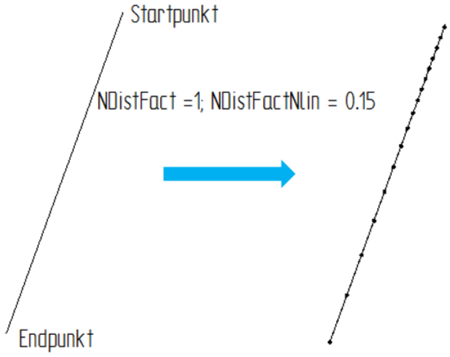
Optional parameters

The following texts may be included. If no information is contained, smartFEM automatically calculates these values from the selected geometry points.

Note:
External radii must correspond to the maximum or minimum distance between the center point and the outer or inner contour of the respective geometry. If the contours are not circular-shaped, it is possible that the radii on the sector lines of the pole segments cannot be determined from the geometry points. In this case, they must be specified.

Keyword	Parameter	Description
Rro =	#.###	Outer radius rotor
Rri =	#.###	Internal radius rotor
Rso =	#.###	Outer radius Stator
Rsi =	#.###	Internal radius Stator
Np = alternative Ns = i.e. Nm =	## ## ##	Number of poles for rotor or stator according to "*Position*" Number of stator slots Number of magnet poles

Node distances or number of segments can be specified as follows for lines and arcs. The text boxes must be positioned in such a way that the anchoring points lie as close as possible to the respective lines or arcs.

NDistFact = or NDist =	#.#### (default: 1.0)	Factor for setting the knot density = Bna * NDistFact = Bna * NDist
NDistFactNlin = or NlinDist =	#.#### (default: 0.0)	Factor for determining a nonlinear node density on the lines. In case of a positive factor and positioning of the text box closer to the end point of the line, the node density is distributed from low to high in the direction from start to end point, in the case of a negative factor or positioning closer to the starting point in the opposite direction. 

NSeg =	###	Number of segments, into which line and circular arcs are divided by the nodes.

Further optional parameters

Keyword	**Parameter**	**Description**
MiddleSlotLine		Marking the line, which divides the slot in the right and left side in unbalanced geometries.
NCoils = NCoilsPerSlot = CoilsLayer =	1 or 2, default = 2 UpDown RingCoils	One or two-layered coil Coil in the left/right slot half Coil in upper/lower slot half (only for two-layered coil) Ring coil around the stator back
NSlotsPerSegment	##	Number of slots per segment in segmented geometries („*SegmentedGeometry*")
NPhases =	##, default = 3	Number of phases
Bna = Bnd =	#.###, default: = 1.0 #.###, default: = 0.3	**B**asic**N**ode**A**ngle = minimum angle between two nodes at the air gap. **B**asic**N**ode**D**istance = minimum distance between two nodes at the air gap.
Dp =	#, default = 2	Number of decimal places for the rounding of the entered and displayed parameter values

With the help of the CAD program *"Solid Edge 2D-Drafting"* from Siemens PLM Software, text can also be assigned to the respective lines, arcs, etc. by means of a text bubble.

7.4.3 Examples of CAD drawings

7.4.3.1 Example of a Stator Geometry

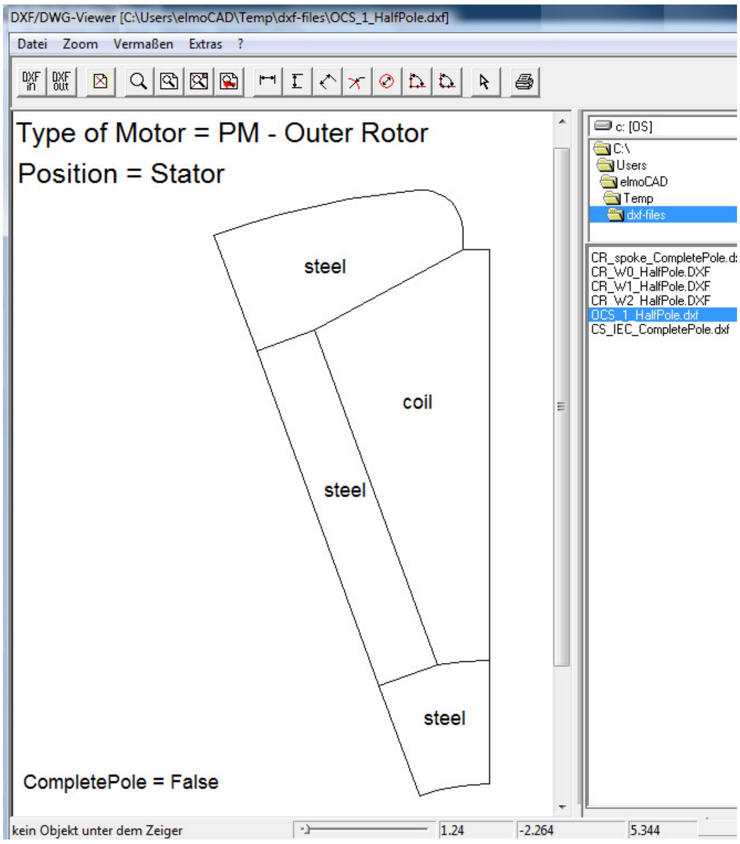


Fig. 254: CAD import - Example stator geometry

smartFEM detects the position in the xy-level and rotates the topology so that it is displayed as the "usual" standard topologies.

7.4.3.2 Example of rotor geometry with node densities

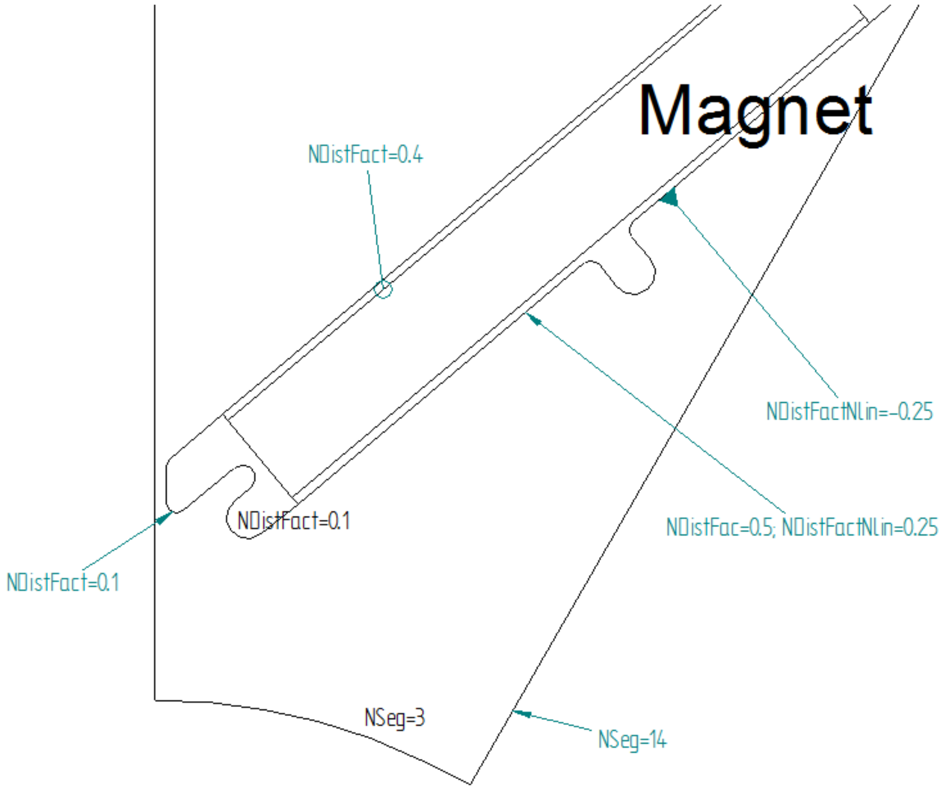


Fig. 255: CAD Import - Example of a rotor geometry

The arrangement of node densities information can be done with the help of text boxes and text bubbles. Currently, only text bubbles in Solid Edge 2D drafting by Siemens PLM software are available and can contain different start/end symbols (e.g. arrow, circle, etc. without bending lines).

7.4.3.3 Example of rotor geometry with *"SPLINE"*

"*Splines*" can be designed in the CAD drawing with n control points and degree < n.

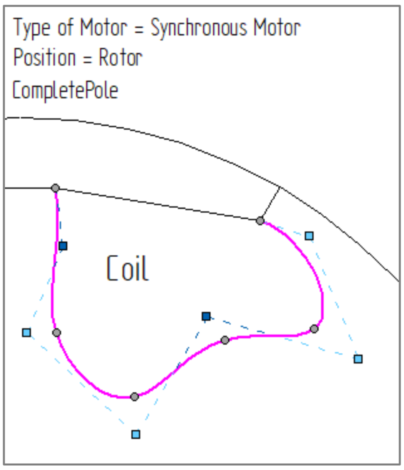
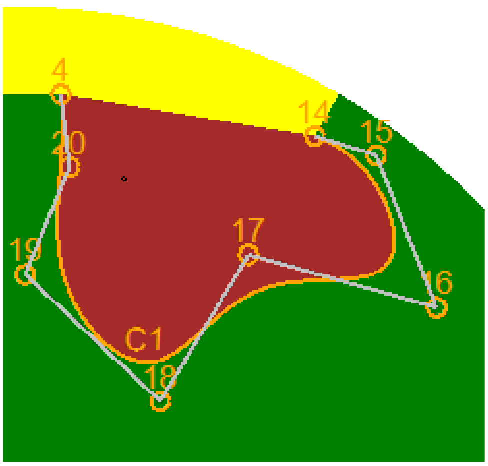


Fig. 256: CAD drawing with SPLINE



smartFEM material view after DXF import of the winding area showing also the control points of the spline which is displayed as curve "C1"

7.4.3.4 Example of a complete stator geometry

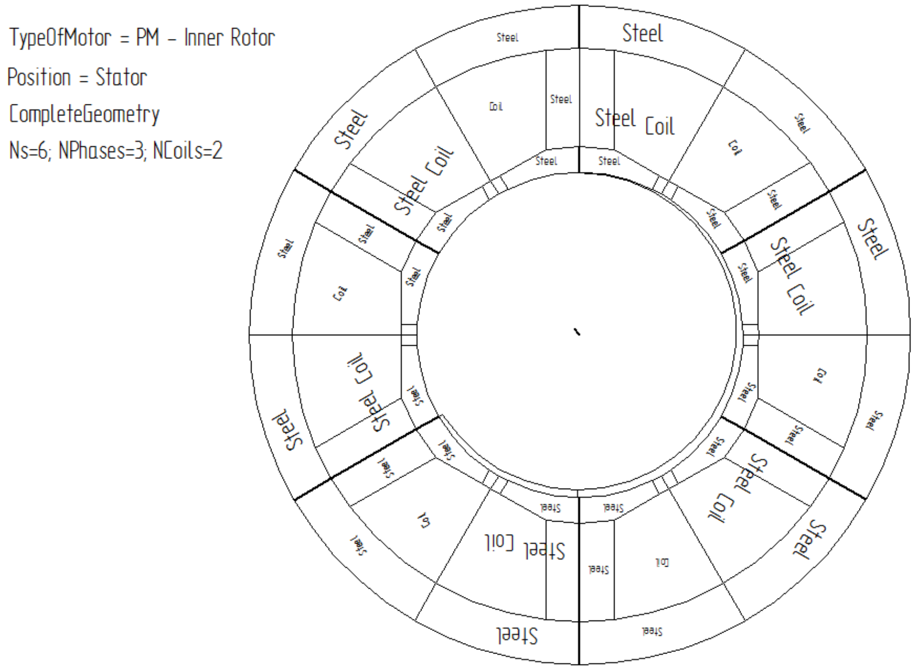


Fig. 257: DXF import - Complete geometry

7.4.3.5 Example of a complete stator geometry with offset

If a line is drawn at the zero point of stator geometry, an eccentricity between rotor and stator is generated in smartFEM according to the length and direction of the line.

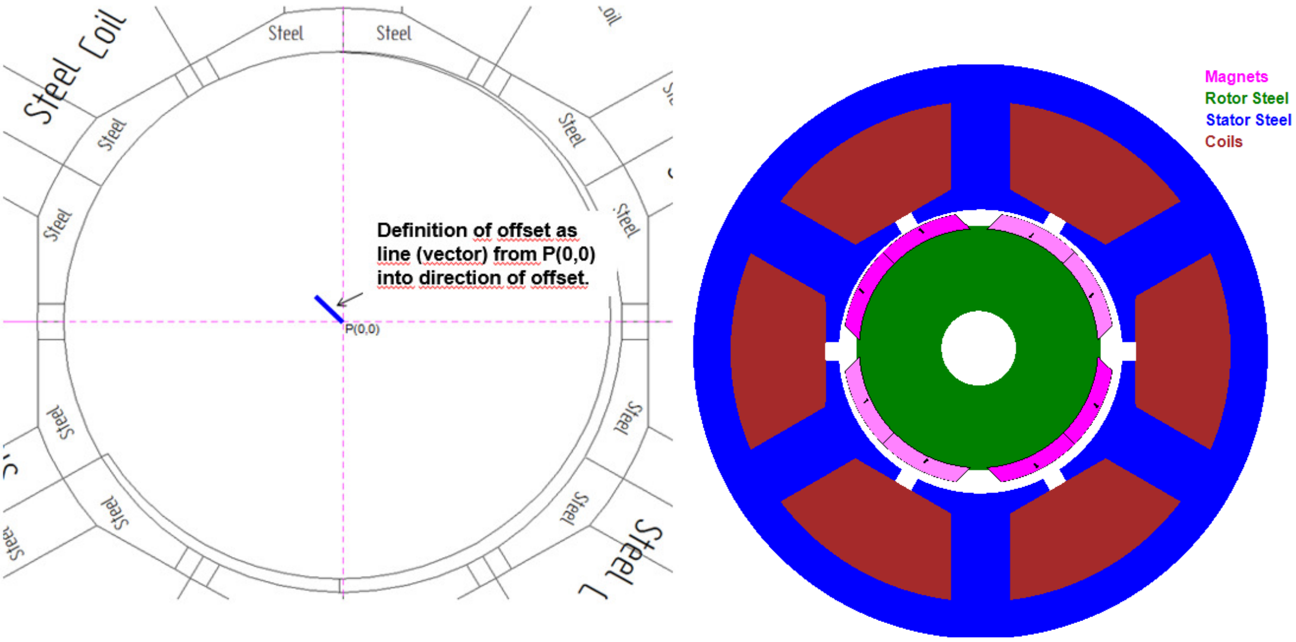


Fig. 258: DXF Import - Eccentricity between rotor and stator

7.4.4 Use of Layers in the CAD Models

If the CAD model should contain more information than the dfx import basics (e.g., guidelines, dimensions, etc.), layers must be used. The drawing objects and text required for the DXF import to smartFEM must be contained in layers that on their part contain the word "smartFEM". Several "smartFEM" layers can be defined. All other information must be drawn in other layers. If no layers are used, the CAD model is allowed to contain only the information required for the smartFEM DXF import.

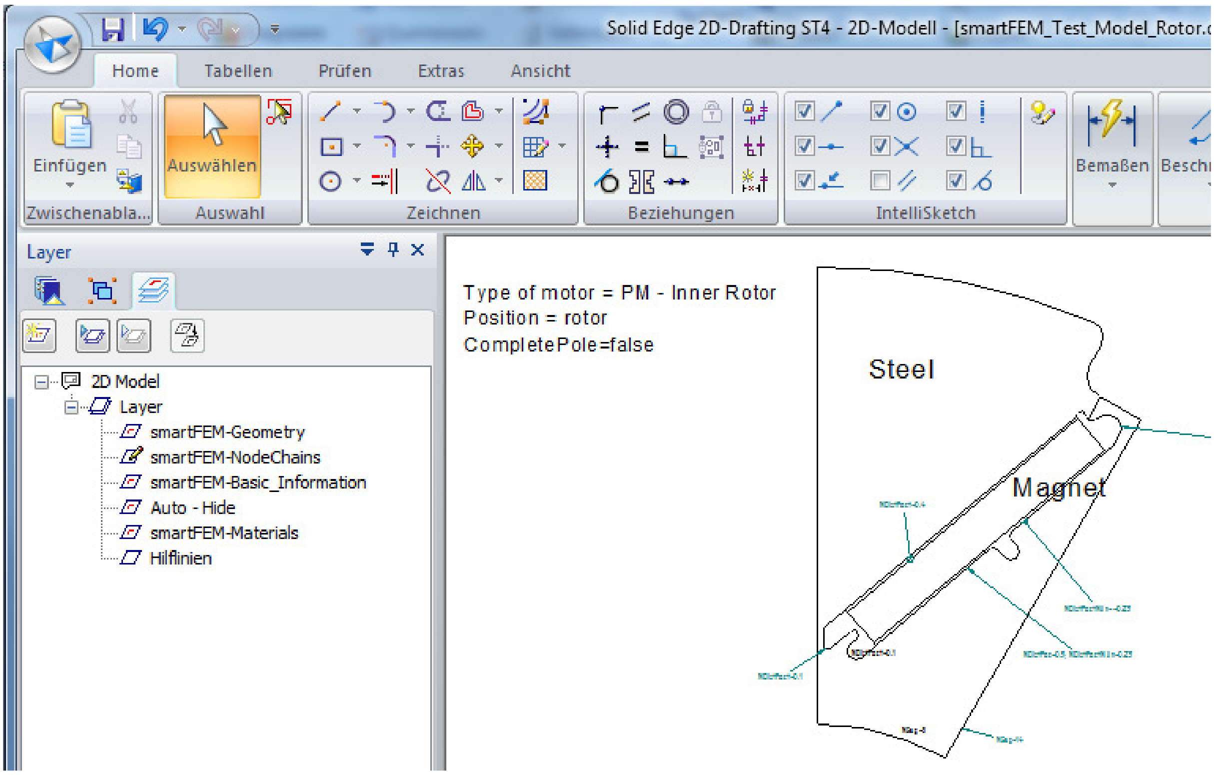


Fig. 259: CAD Import - Using Layers

7.4.5 Keywords and Parameter of Linear Motors

Topologies of linear motors contains always out of the rotor and stator geometry. By definition must the rotor geometry be below the stator and airgap geometries. For identification of geometry and motor parameter must different information be defined as *“Basic Parameter”* in the DXF drawing. The anchor points of the text boxes must be positioned very close to the related points.

Obligatory Keywords and Parameter

Keyword	Parameter	Description
Np	= #	Number of poles within the active motor length.
NpComplete	= #	Total number of poles.
AirGapTickness	= P_1 : P_2	Geometry points which indicates the thickness of the airgap .
MotorStartsXCoord	= P_3	Geometry point which indicates the x-coordinate of the first left point of the motor geometry.
SlotStartsXCorrd	= P_4	Geometry point which indicates the x-coordinate of the first left point of the first left stator pole.
SlotPitchLength	= P_3 : P_4	Geometry points which indicate the length of a stator pole
MotorLength	= P_5 : P_7	Geometry points which indicate the active length of the motor.

Example:

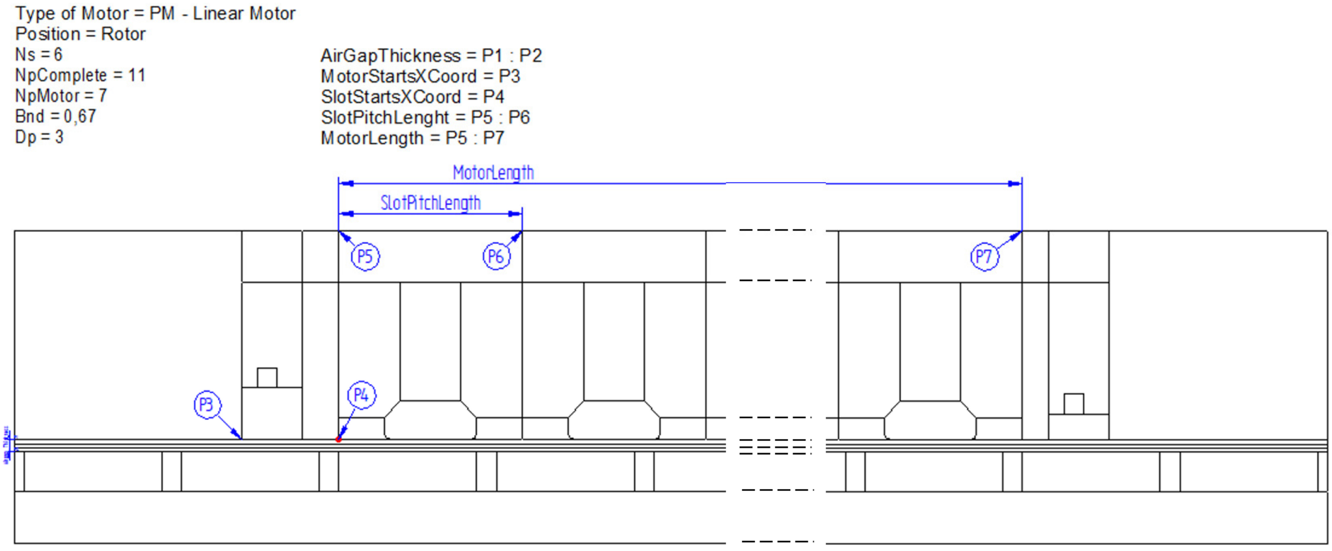


Fig. 260: DXF-Import Linear Motor - Basic Parameter

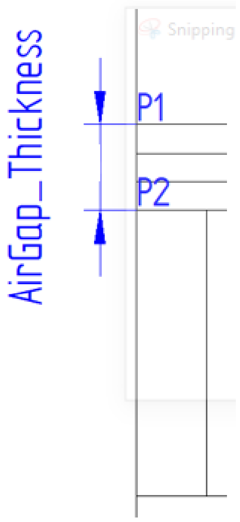


Fig. 261: DXF-Import Linear Motor - Basic Parameter "AirGapThickness"

7.4.5.1 Keywords and Parameter of Actuators

Keyword	Parameter	Description
Type of Motor	= Actuator	Type of model
CoordinateSystem	= [xy | rz] (xy=default)	Coordinatesystem xy or rotationsymmetric
NPhases	= #	Number of phases
Ns	= #	Number of slots
NCoilsPerSlot	= #	Number of coils per slot
CompletePole or HalfPole	[= False | True]	complete or half geometry (madatory for COOrdinateSystem = rz)
ForcePath		Polyline indicating the path to calculate the forces applied to the platen.
MoveVector		Line indicating the direction and length of the movement.of the platen.

Fig. 262: DX-Import Actuator - Basic Parametrer

Example:
(see also § 6.7 Actuator)

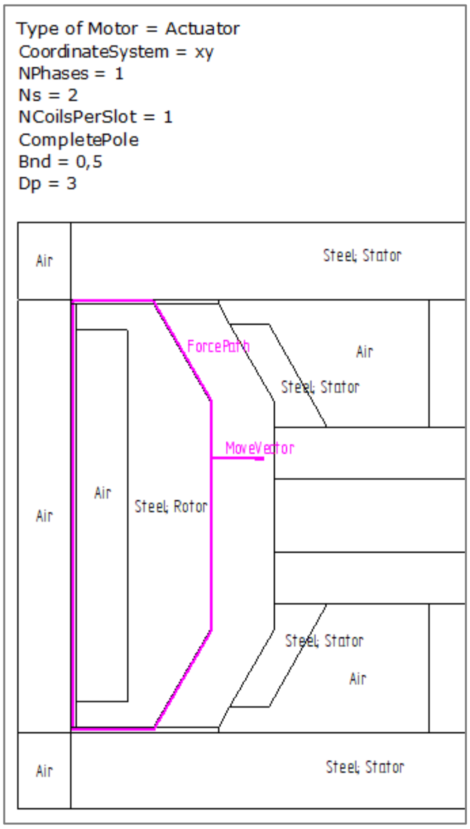


Fig. 263: DXF-Import Actuator - ForcePath and MoveVector

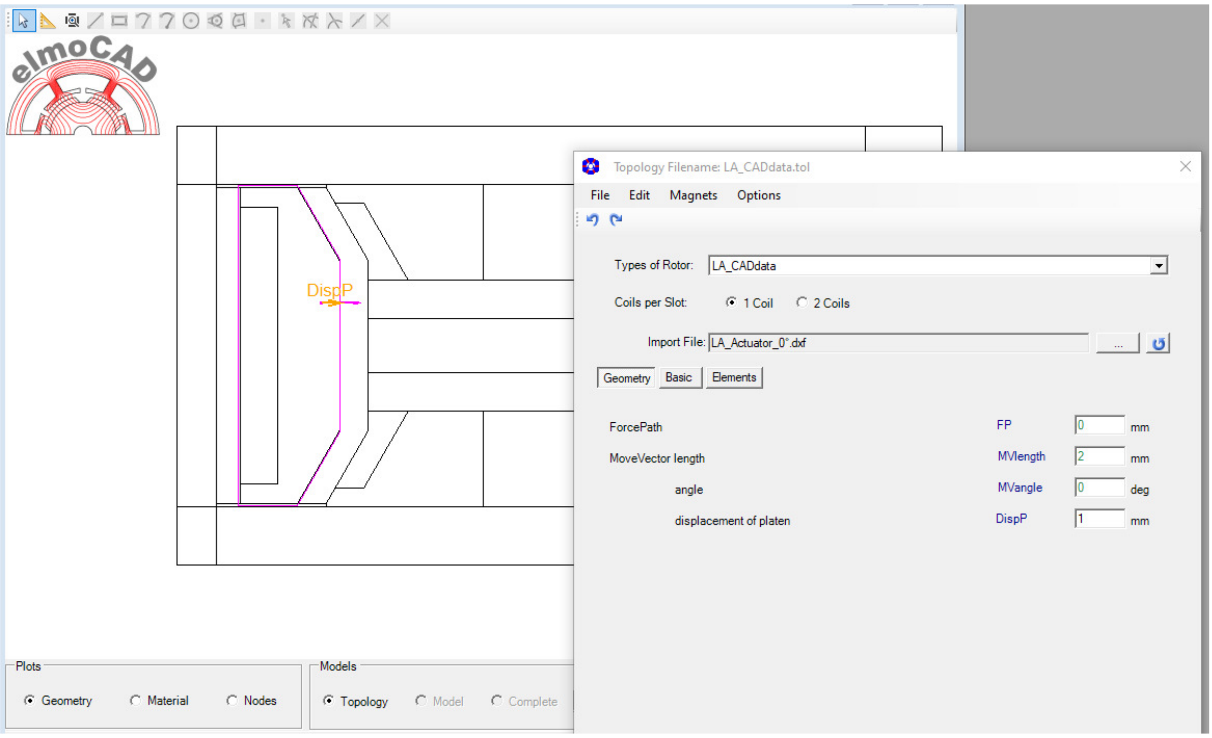


Fig. 264: DXF-Iport Actuator - ForcePath and MoveVector in smartFEM (Geometry Plot)

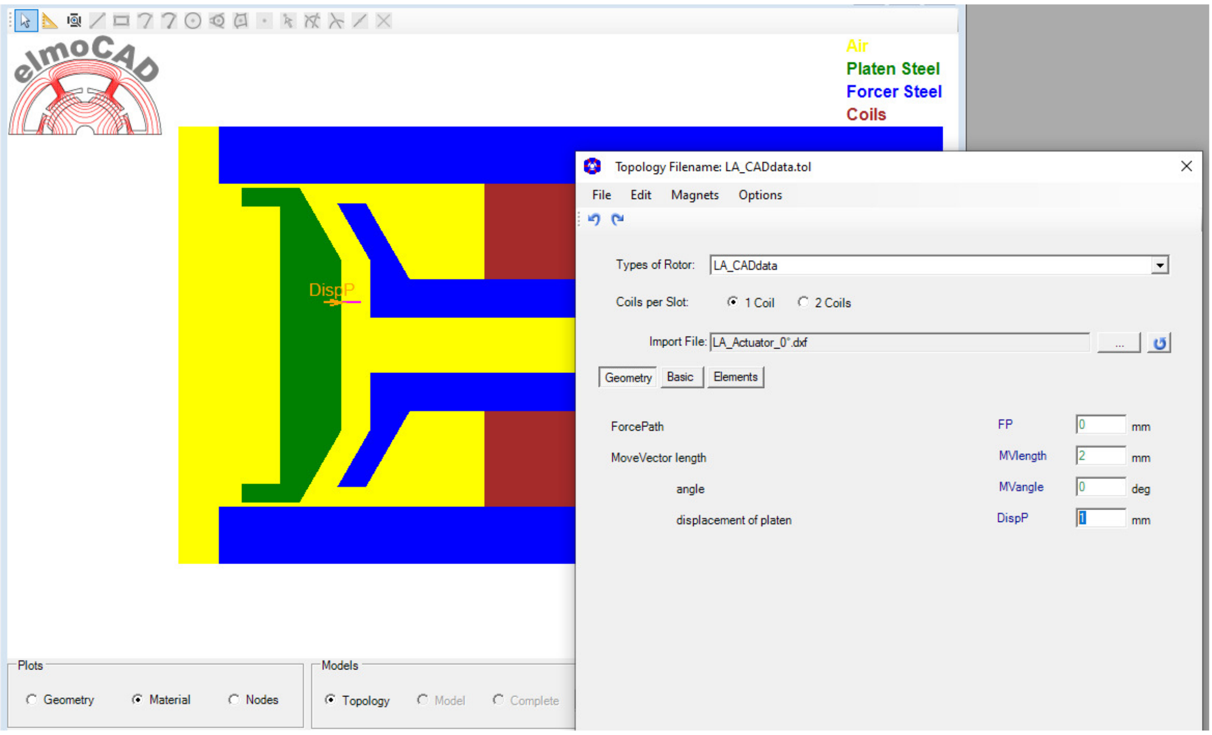


Fig. 265: DXF-Import Actuator - ForcePath and MoveVector in smartFEM (Material Plot)

After DXF import is the geometry of the actuator displayed as in the CAD drawing.

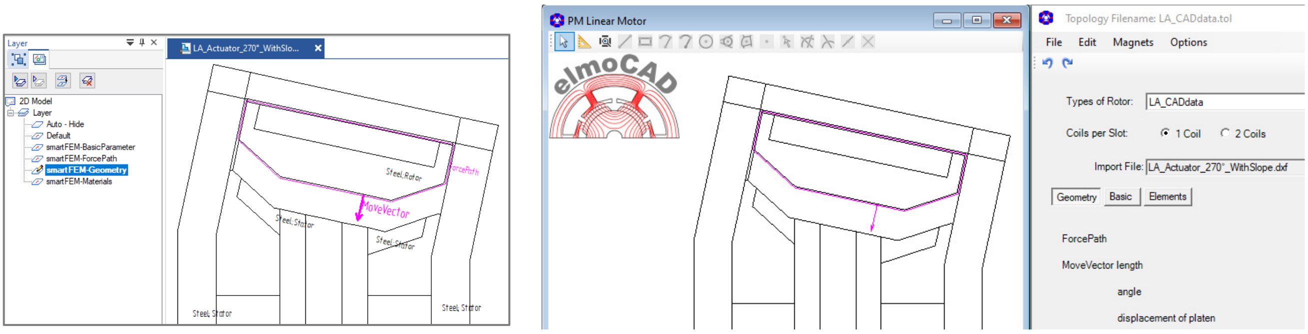


Fig. 266: DXF Import Actuator - Direction of the "MoveVector" ≠ 0

After *"Apply"* is the geometry automatically rotated in such a way that the *"MoveVector"* is in 0°mech (x-axis) direction. With that can the ration of x- and y-forces properly be calculated (x-direction = moving direction).

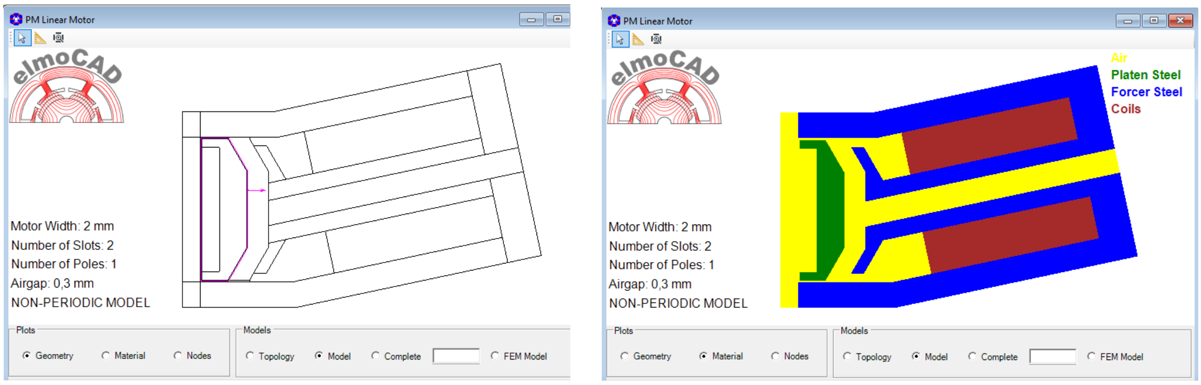


Fig. 267: DXF Import Actuator - Rotated geometry with "MoveVector" in direction of the x-axis

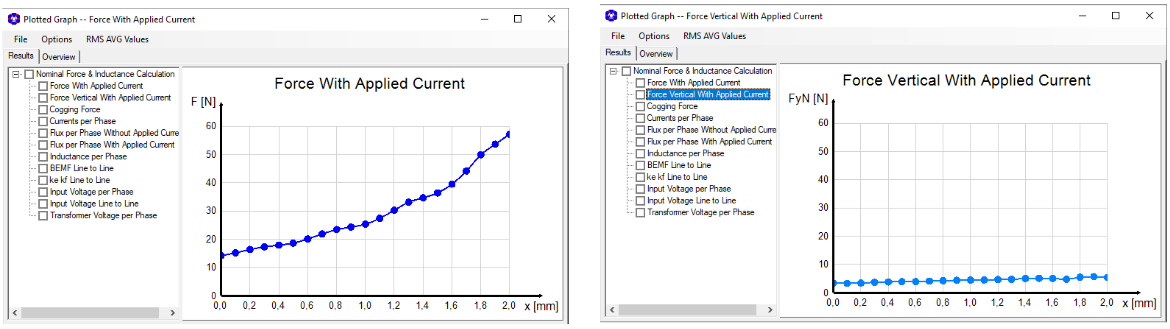


Fig. 268: DXF Import Actuator - Forces in x- and y-direction when geometry is asymmetrical

7.4.6 Functional description of the DXF import

After selecting the CAD_Rotor or CAD_Stator topology, an empty geometry and parameter window is displayed as well as a selection window for selecting a DXF or CSV file.

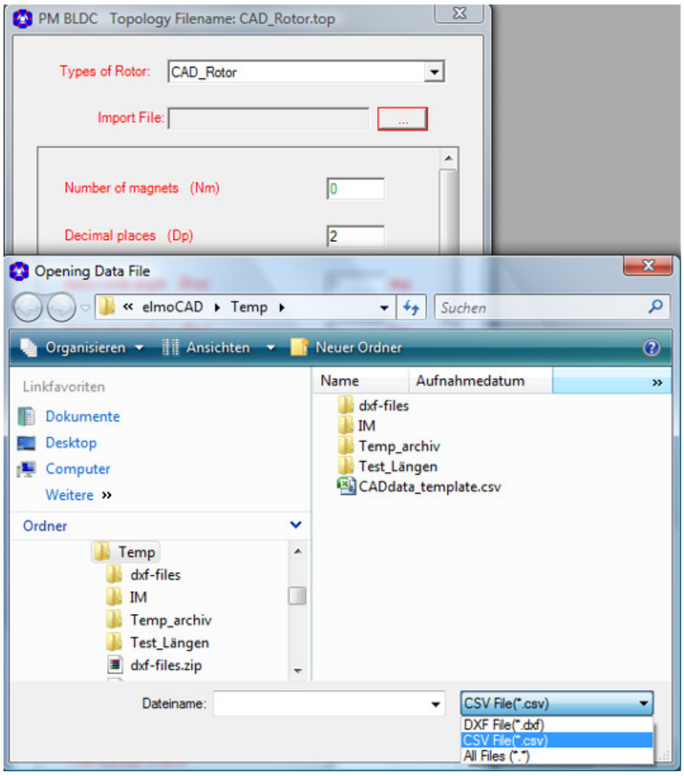


Fig. 269: DXF-Import - Selection of the DXF or CSV file

After selecting the corresponding file, the import is performed toherther with the construction of geomerty and the display of the parameter list:

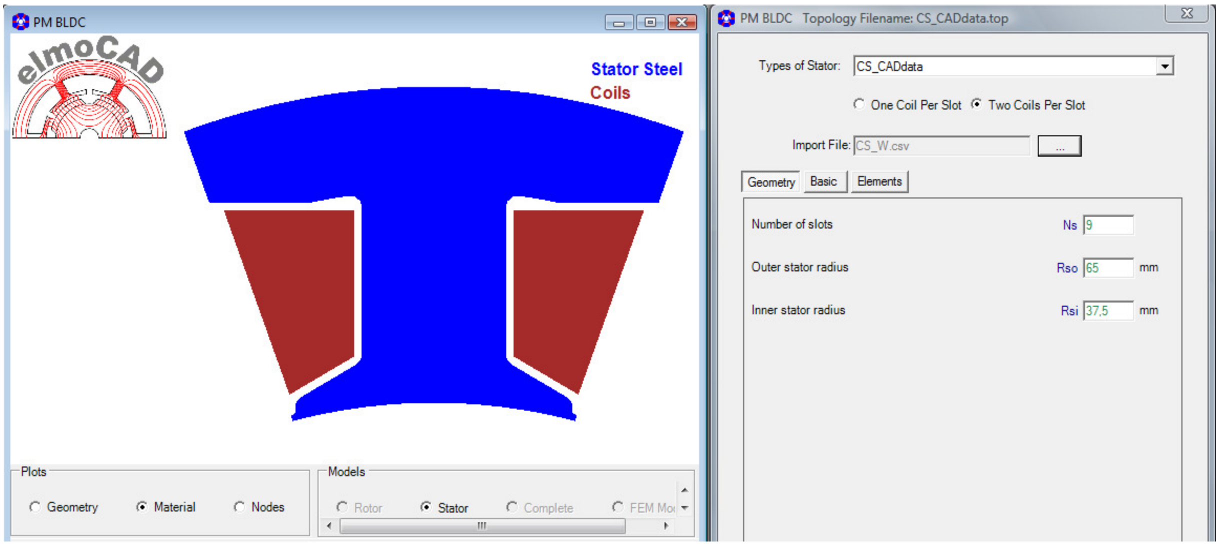


Fig. 270: DXF-Import - Example of imported stator geometry

The information about individual lines, arcs, etc. can be specified by entering their index number. In addition to the numeric information in the parameter list, context-sensitive auxiliary information is displayed in the graphics window.

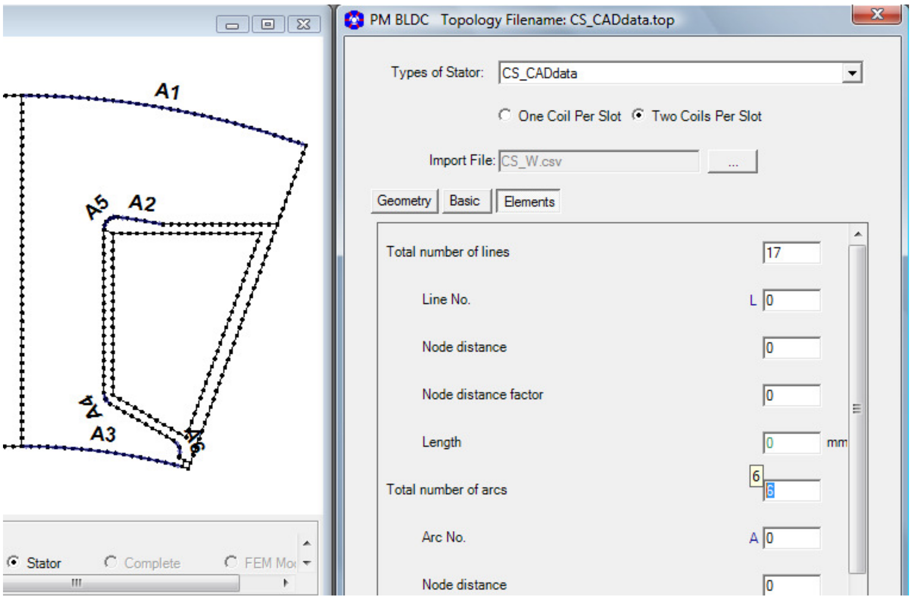


Fig. 271: DXF-Import - Information on geometry elements

Entering the line, arc, area and point numbers displays additional information. The node distances can be changed for each element.

Example:

- Bow: A5
- Center point: 16
- Starting point: 18
- Endpoint: 17
- Factor for minimum Node distance: 0.5 (Node distance = factor * basic node distance)
- Radius: 1 mm
- Opening angle: 97.73 °

In the "Geometry" and "Nodes" view of the graphics window, individual elements can also be selected with a right mouse click on the corresponding element.

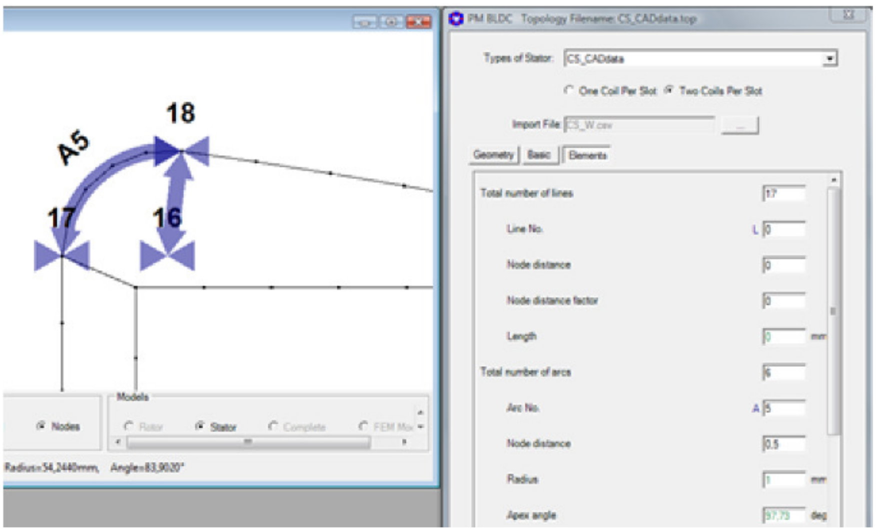


Fig. 272: DXF-Import - Selection of a geometry element

7.4.7 Pre-Requirements and Error Notes

When creating DXF or CSV files, the following points have to be considered, so that smartFEM can build up a simulation-capable geometry model.

7.4.7.1 Free Area Elements

The geometries must not contain any surface elements (islands) that are not connected to another area element by a line or circular arc.

If this is still the case, the corresponding surface element is displayed with a red border in the geometry representation, an error message is displayed and the "Bad Geometry" button is displayed instead of "Apply". A connection **must** then be established in the CAD system and the import performed again.

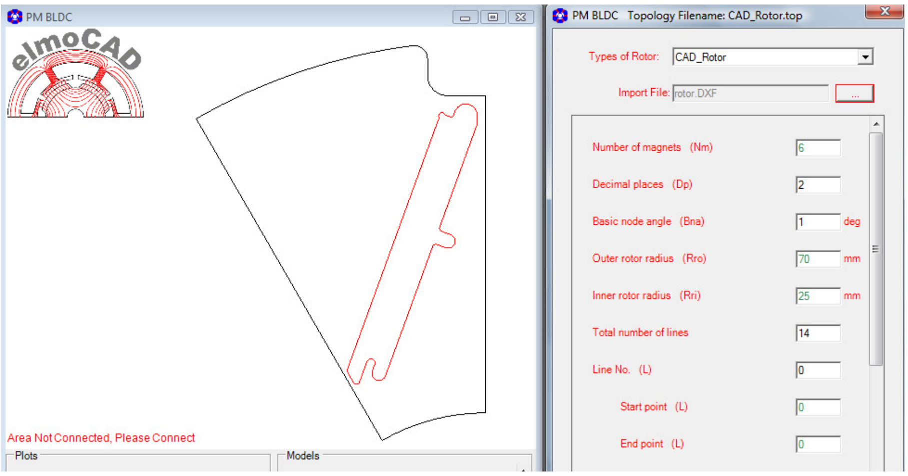


Fig. 273: DXF-Import - Free surface elements

7.4.7.2 Unique Points

Rounding differences in the creation of the DXF or CSV files for the import can lead to the possible situations that the same points have slight different xy-coordinates and are recognized as two points. To prevent this, the minimum distance between two points „*MinDist*“ can be specified as a parameter in the parameter group *"Basic".* xy-coordinates which have a smaller distance to each other are then recognized as one point.

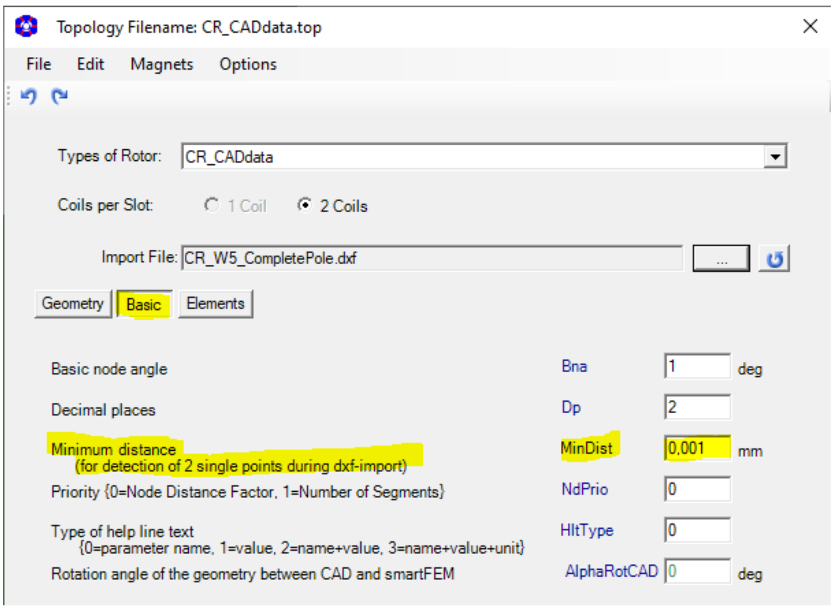


Fig. 274: DXF Import - Minimum Distance of Points

7.5 DXF Export

The geometries created with smartFEM can be exported in dxf format in two different ways, in order to be used with other programs:

- Export from the respective topology window

  A complete geometry model including all related information as text for e.g. CAD systems for changes to geometry and/or text parameters with subsequent re-import into smartFEM

- Export via the smartFEM main menu

  Various types of geometry model for e.g. CNC machines for sheet metal cutting

7.5.1 Export from the respective topology window

This function is called up via the menu *"File - Export Topology to DXF"* specified in the topology window.

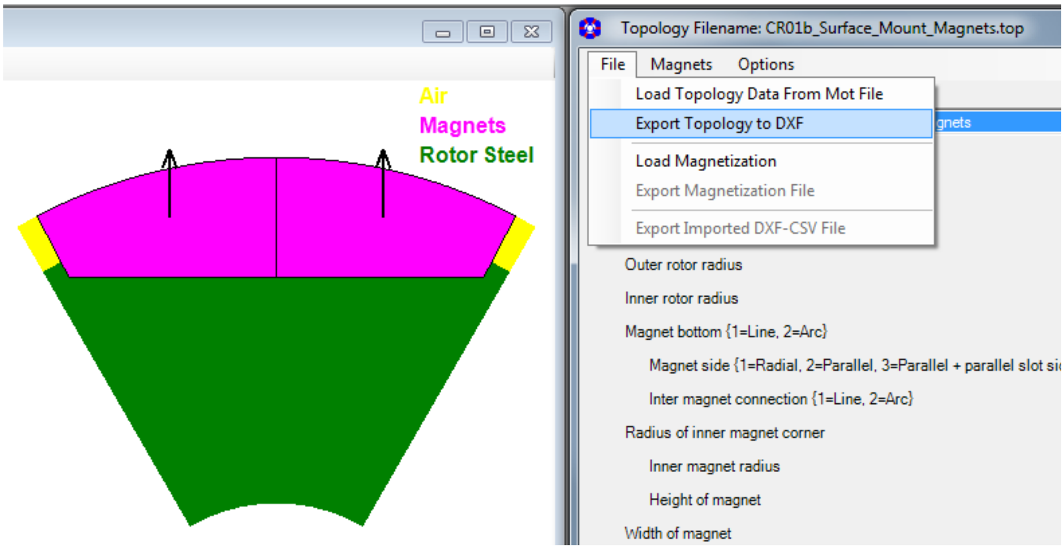


Fig. 275: DXF Export - export topology to DXF

The user can specify in which directory the DXF file should be stored. Different layers with the term "smartFEM ..." are created for the respective information types. Text size of the of node distances are adjusted to the respective lengths of the lines and arcs.

After being opened in a CAD system, geometry and text parameters can be modified and, after saving as a DXF file, imported back into smartFEM with the corresponding CADdata topology (in this example CR_CADdata).

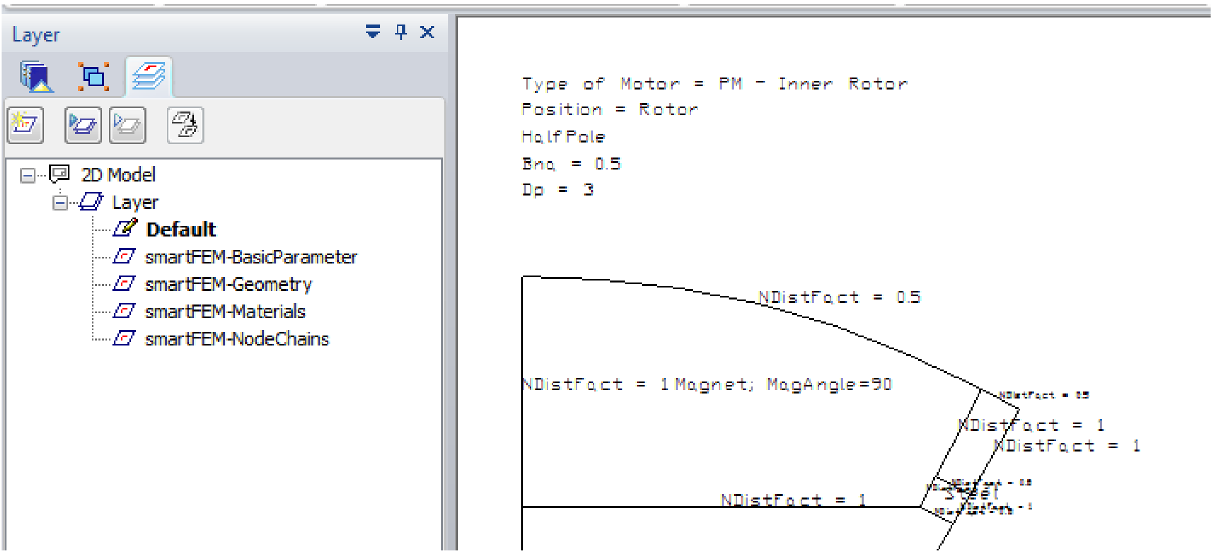


Fig. 276: DXF Export - representation in a CAD system

The 2D-Model of the CAD system can contain any drawing and text information. The information required for the import into smartFEM and only these must respectively can be contained in layers that contain the text "smartFEM".

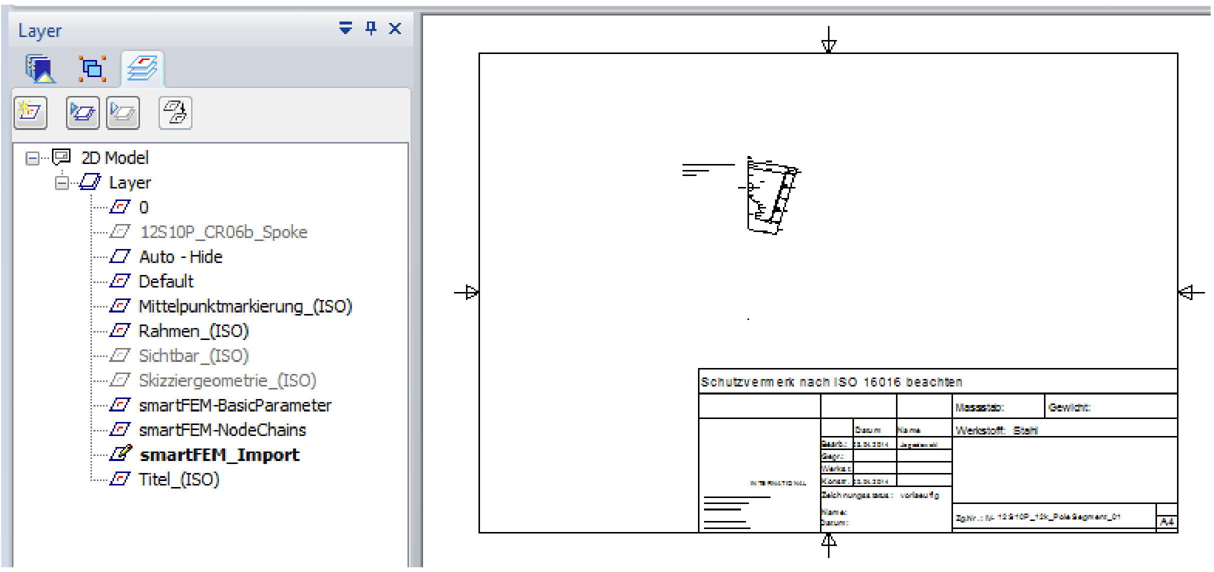


Fig. 277: DXF Export - smartFEM layer in 2D model of the CAD system

7.3.2 Export via the smartFEM main menu

With this function, only geometric data is exported to a DXF file in the smartFEM geometry file according to the respective view:

- Rotor
- Stator
- Periodic model
- FEM model

additionaily, e.g. for the programming of CNC machines

- Export Material Contours
- Export Steel Material Contours

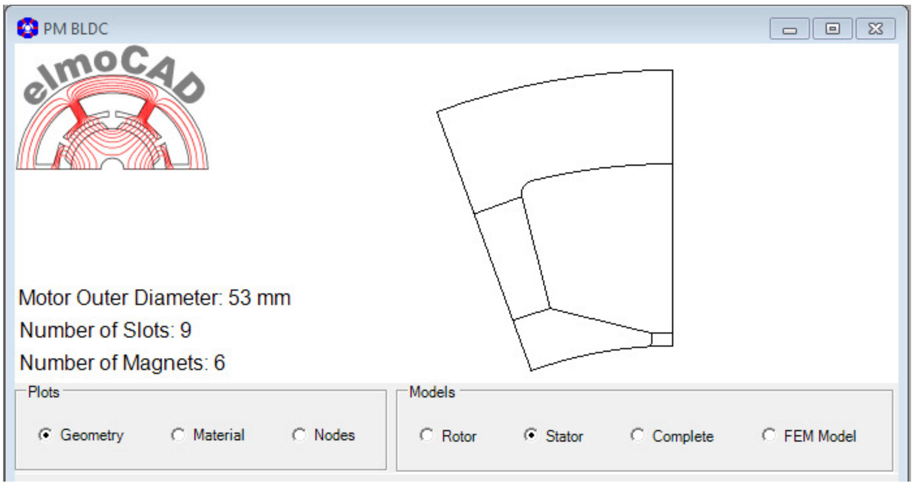


Fig. 278: DXF Export - Example stator geometry

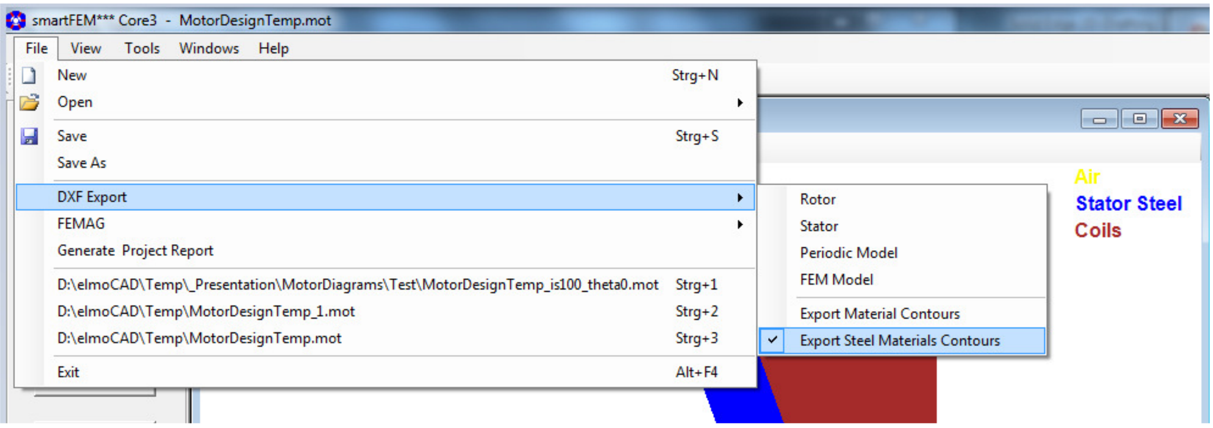


Fig. 279: CAD Export - Selection of different types

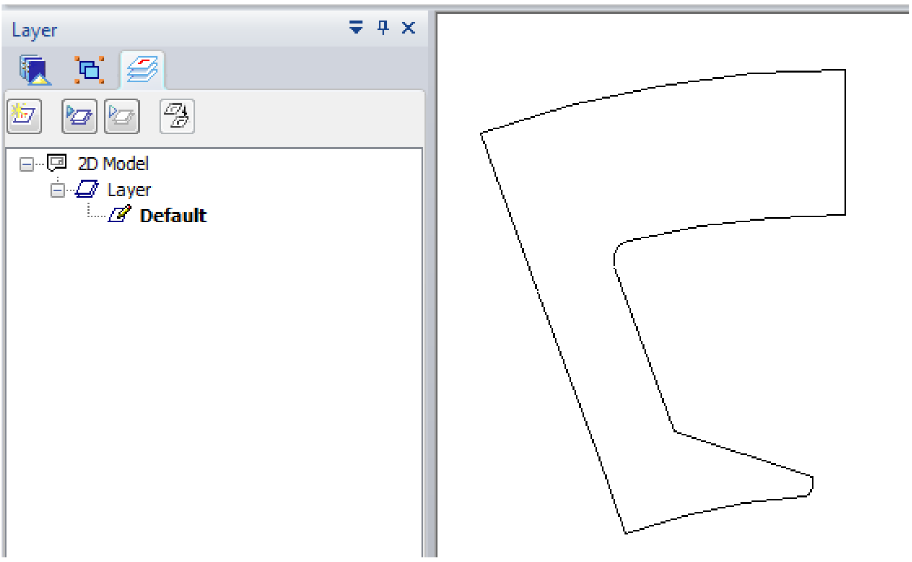


Fig. 280: DXF Export - Example: Contour of the stator laminated core

To export the entire model, the winding editor must be opened by clicking on the "Winding Defined" button. The entire model is then displayed and can be exported with *"File - DXF Export - Periodic Model".* Uneccessary drawing lines need to be removed in CAD system, so that only e.g. the contours can be further processed.

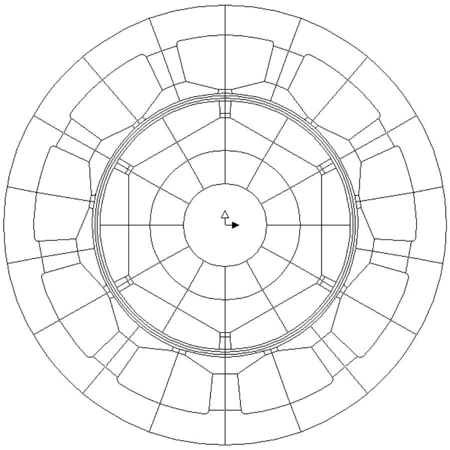


Fig. 281: DXF Export - Entire machine model

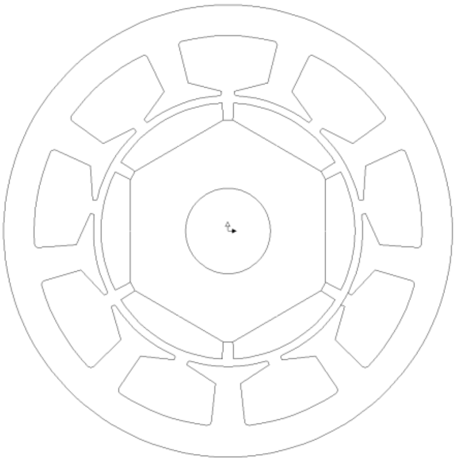


Fig. 282: DXF Export - Material contours

7.6 CASPOC

For simulation of motor control with the simulation tool for power electronics *CASPOC*[1], the parameters required for Ld/Lq calculation can be generated directly via the *"Data- Caspoc - Add Sets"* dialog and then calculated using *"Solve All".*

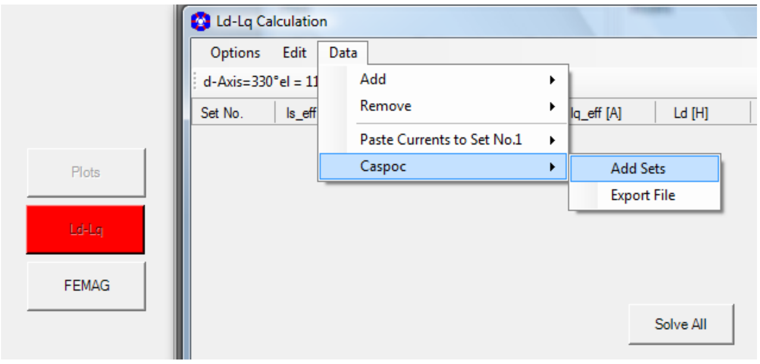


Fig. 283: CASPOC calculation parameters

Subsequently, the generated result table with further results data can be stored in a XML formated file which can be processed directly by CASPOC.

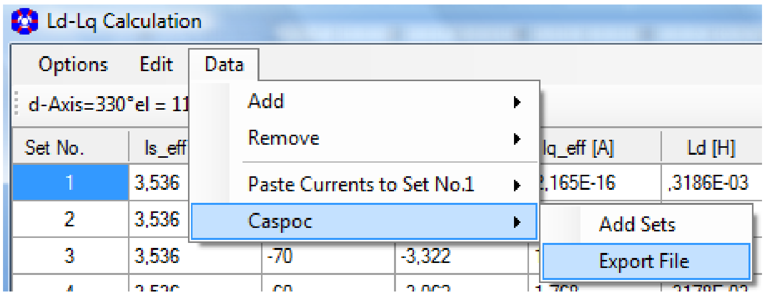


Fig. 284: CASPOC - Saving of the results data to an export file

The file contains the following data:

Motor parameters
- stator inductance Ld
- stator inductance Lq
- Winding resistance
- moment of inertia of the rotor
- number of pole pairs

Data table1
- Rotor position [° mech]
- Phase shift current [° el] (not used = 0)
- Amplitude of current (not used = 0)
- Torque (= locking torque)
- ke1 Voltage constant phase U
- ke2 Voltage constant phase V
- ke3 voltage constant phase W
- flux linkage 1
- flux linkage 2
- flux linkage 3

[1] CASPOC is a product of Simulation Research, NL - Alphen aan den Rijn

Data table2

- Rotor position [° mech] (=0)
- Phase shift current [° el]
- Amplitude of the current
- Torque
- ke1 voltage constant phase U (not used = 0)
- ke2 voltage constant phase V (not used = 0)
- ke3 voltage constant phase W (not used = 0)
- Ld
- Lq
- flux linkage 3 (not used = 0)

	A	B	C	D	E	F	G	H	I
1	name	name2	value_si	description	name3	name4	lookupdata row 1	lookupdata row 2	lookupdata row 3
2	1	Ld	0,000321801	Stator inductance d Axis					
3	2	Lq	0,000422774	Stator inductance q Axis					
4	3	Rs	0,155299601	Winding Resistance					
5	4	J	0,0001	Rotor Inertia					
6	5	p	3	Number of Pole Pairs					
7					0	Rotor Position(°mech)	0	0,017453293	0,034906585
8					1	Theta Current(°el) not used set to zero	0	0	0
9					2	Current Amplitude(A) not used set to zero	0	0	0
10					3	Temperature(Celcius)	20	20	20
11					4	Cogging Torque(Nm)	0,007046272	-0,006898098	-0,02323248
12					5	Ke1(Vs/rad)	-0,002313303	-0,002547804	-0,002780866
13					6	Ke2[Vs/rad]	0,00464771	0,004647788	0,004647385
14					7	Ke3[Vs/rad]	-0,00231503	-0,002080147	-0,00184546
15					8	FluxLinkage1(Vs)	0,013270199	0,012865044	0,012420811
16					9	FluxLinkage2(Vs)	-8,72E-07	0,000773758	0,001548392
17					10	FluxLinkage3(Vs)	-0,013269248	-0,013635474	-0,013962494
18					0	Rotor Position(°mech) set to zero	0	0	0
19					1	Theta Current(°el)	1,745329252	1,745329252	2,268928028
20					2	Current Amplitude(A)	20	10	10
21					3	Temperature(Celcius)	20	20	20
22					4	Torque(Nm)	1,38	0,686	0,551
23					5	Ke1(Vs/rad) not used set to zero	0	0	0
24					6	Ke2(Vs/rad) not used set to zero	0	0	0
25					7	Ke3(Vs/rad) not used set to zero	0	0	0
26					8	Ld(H) FluxLinkage1(Vs) used for Ld	0,000336764	0,000321772	0,00031937
27					9	Lq(H) FluxLinkage2(Vs) used for Lq	0,000414097	0,000423871	0,00042504
28					10	FluxLinkage3(Vs) not used set ot zero	0	0	0

Fig. 285: CASPOC - example XML file

8 Notes