In The Name of God The Most Compassionate, The Most Merciful

Brushless PM Machines

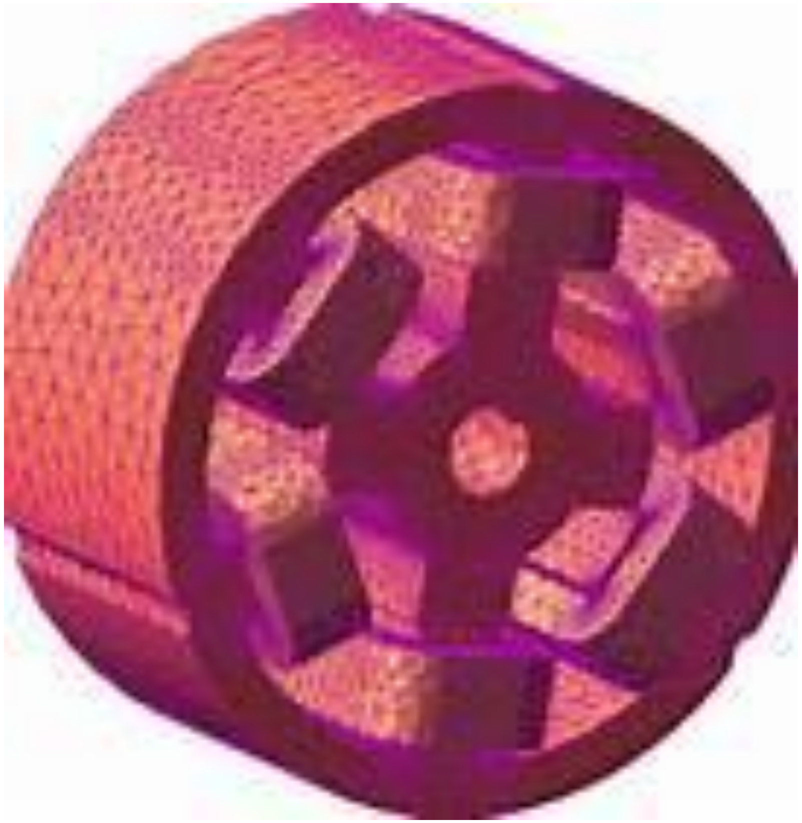
Design, Optimization and Analysis

Table of Contents

Numerical Techniques

The most widely used techniques to numerically solve partial differential equations (PDEs) are:

- **Finite element method (FEM)**
- **Finite difference method (FDM)**
- **Finite volume method (FVM)**



Software

Famous software packages for electromagnetic problems:

- **ANSYS®** (Electromagnetic)
- **Ansoft Maxwell®** (ANSYS Maxwell from 2008)
- **COMSOL Multiphysics®**
- **CEDRAT®**
  - FLUX® 2D
  - FLUX® 3D

ANSYS Ansoft Maxwell®

- Ansoft Maxwell (which is now known as ANSYS Maxwell) is a program that can be used to **visualize magnetic fields** and **predict magnetic forces**, **torques**, **inductances**, **back-emf** and other crucial electromagnetic quantities.

- Electromagnetic problems are **difficult** to be accurately solved because the materials used are **nonlinear** and the fields **cannot be confined** like electrons within wires.

- This program makes the design and analysis of electromagnetic devices much **easier**.

ANSYS Ansoft Maxwell®

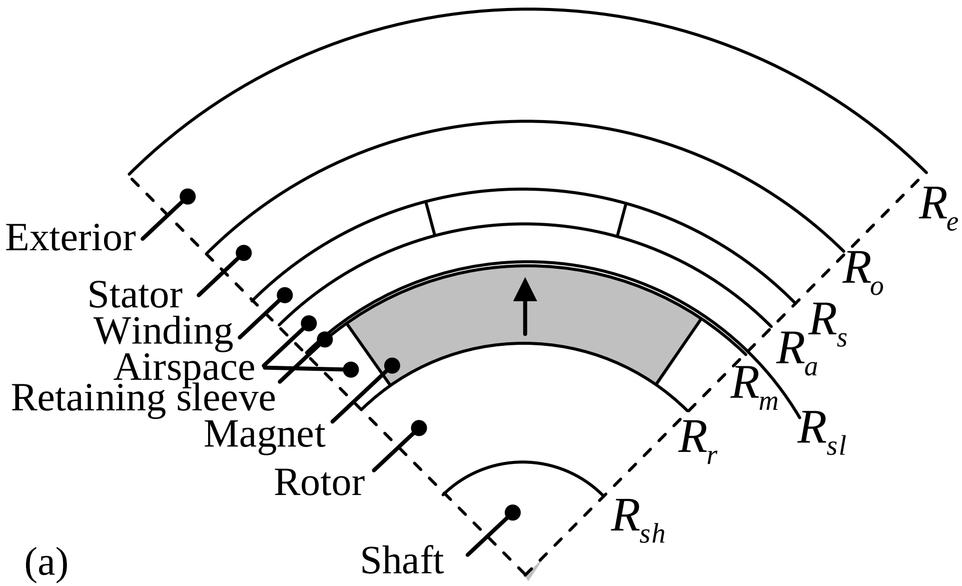
Following steps describe the solution of an electromagnetic problem using Ansoft Maxwell:

1. Set up a **project** (define the file name and type of problem)
2. Define the **model** (draw the geometry of the problem or import the file from drawing package such as AutoCAD)
3. Assign the **material properties** or select the material from the library (specify what each part of the system is made of)
4. Impose **boundaries**
5. Apply **excitation sources** (currents flowing in any part)
6. Define **executive parameters** (e.g. for force or torque calculations)
7. Select the **solution options** (to compromise between accuracy and computational speed)
8. **Solve** the problem (automatic mesh and refining the mesh)
9. **Postprocessing**

ANSYS Ansoft Maxwell®

- The best way to learn a software is through **examples**.
- The first example is to analyze the 2D magnetic flux distribution of a **slotless brushless PM machine** with the following characteristics:

Variables		parameters
p	number of pole-pairs	5
α_p	Pole arc /pole pitch	0.70
μ_r^m		1.05
$\mu_r^s = \mu_r^r$		600
$\mu_r^{sl} = \mu_r^{sh}$		1
B_{rem}	(T)	1.00
I_m	(A)	10
R_{sh}	(mm)	20.0
R_r	(mm)	46.5
R_m	(mm)	59.5
R_{sl}	(mm)	60.0
R_a	(mm)	60.5
R_s	(mm)	64.0
R_o	(mm)	70.0
L	(mm)	75.6
ω	(rad/s)	157
N_c	Turns per coil	15



(a)

ANSYS Ansoft Maxwell®

Step 1: Set up a **project**

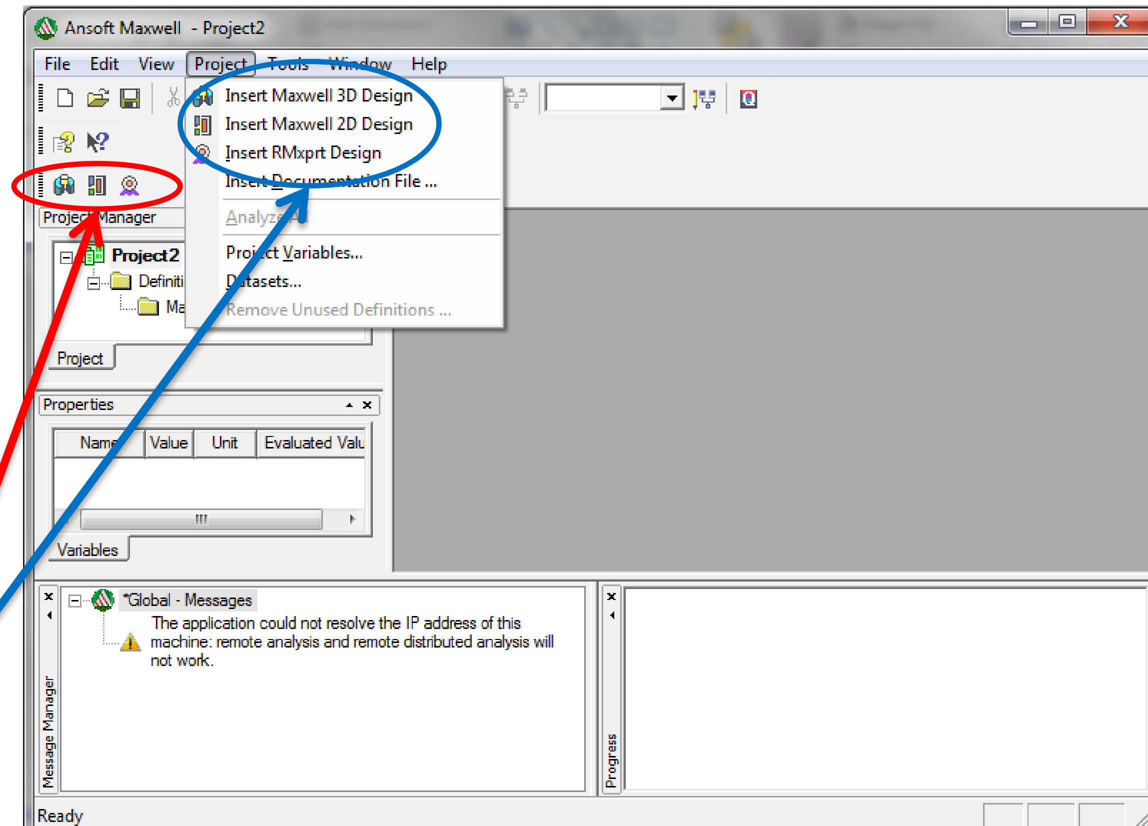
There are three options to start the modeling:

1- Maxwell 2D Design

2- Maxwell 3D Design

3- RMxprt Design

Which can be selected from **shortcut buttons** or **Project** menu

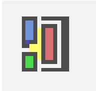


ANSYS Ansoft Maxwell®

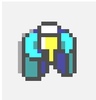
Step 1: Set up a **project**

The applications of these three options are:

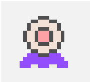
1- **Maxwell 2D Design**: for two-dimensional analysis from scratch.

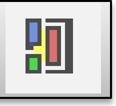
 *Project > Insert Maxwell 2D Design*

2- **Maxwell 3D Design**: for three-dimensional analysis from scratch.

 *Project > Insert Maxwell 3D Design*

3- **RMxprt Design**: for some available electromagnetic devices.

 *Project > Insert RMxprt Design*

We will select Maxwell 2D Design by clicking on 

ANSYS Ansoft Maxwell®

Step 1: Set up a **project**

Some of the electromagnetic devices via **RMxprt Design**

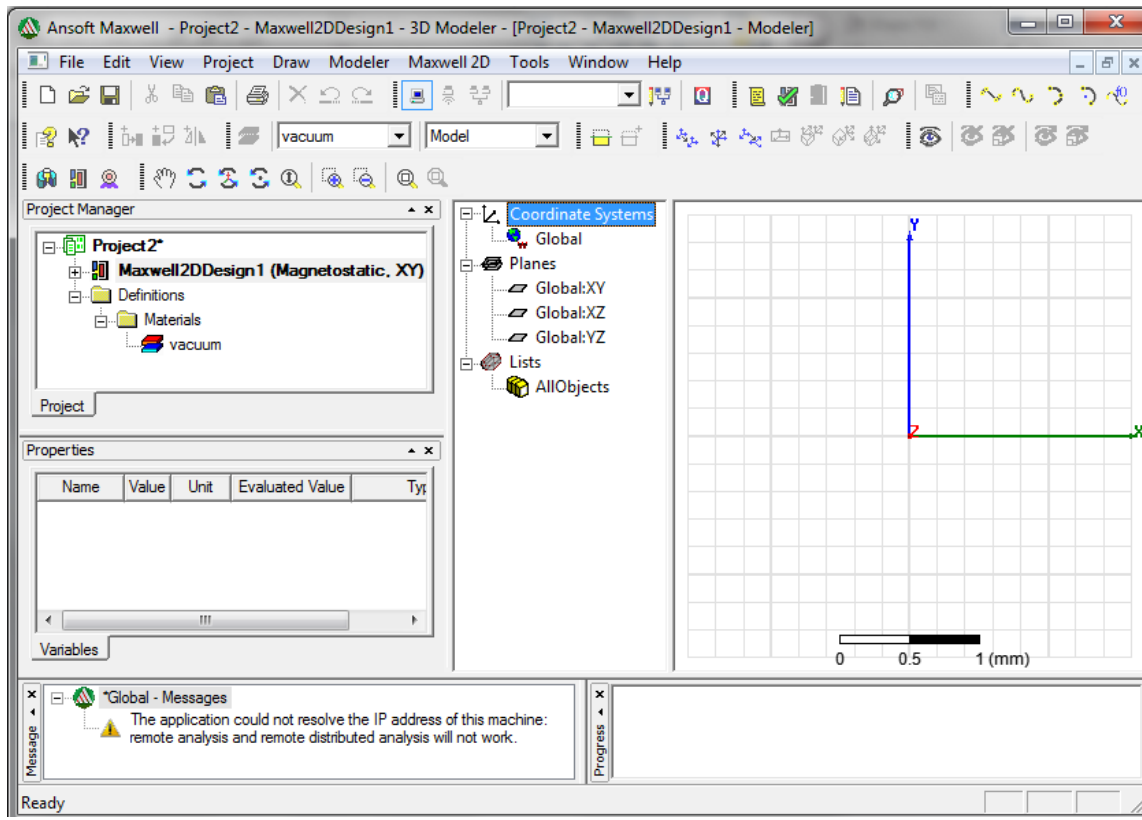
- **Three Phase Induction Motor**
- **Single Phase Induction Motor**
- **Three Phase Synchronous Machine**
- **Brushless Permanent-Magnet DC Motor**
- **Adjust-Speed Synchronous Machine**
- **Permanent-Magnet DC Motor**
- **Switched Reluctance Motor**
- **Line-Start PM Synchronous Motor**
- **Universal Motor**
- **DC Machine**
- **Claw-Pole Synchronous Machine**
- **Three Phase Non-Salient Synchronous Machine**
- **Generic Rotating Machines**

ANSYS Ansoft Maxwell®

Step 1: Set up a **project**

The software has several panels:

- **Project manager**
- **Properties**
- **Message manager**
- **Progress**
- **Modeler window**



ANSYS Ansoft Maxwell®

Step 1: Set up a **project**

- **Project manager** contains a design tree to list the structure of the project
- **Properties** for changing model parameters and attributes
- **Message manager** to view any error or warning occurs
- **Progress** to view the solution progress
- **Modeler window** contains the model and model tree

ANSYS Ansoft Maxwell®

Step 1: Set up a **project**

- In the project manager, right-click on the **Maxwell2DDesign** and select **Rename** to change the name.
- In the project manager, right-click on the **Maxwell2DDesign** and select **Solution Type** to change the **Geometry Mode** (Cartesian or cylindrical) and also to select

Magnetic:

Magnetostatic

Eddy Current

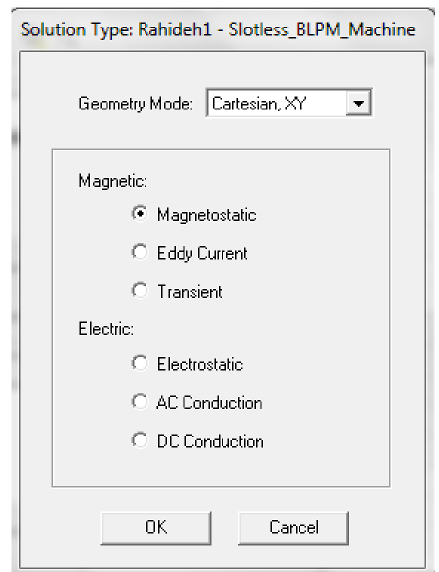
Transient

Electric:

Electrostatic

AC Conduction

DC Conduction



ANSYS Ansoft Maxwell®

Step 2: Define the **model**

- The model can be imported from other packages via:

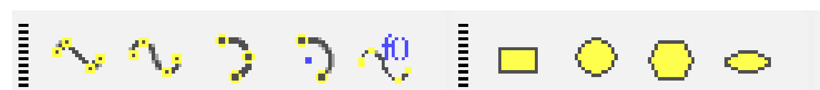
  Modeler > Import…

  Or

- Using the drawing facilities of the ANSYS Maxwell such as

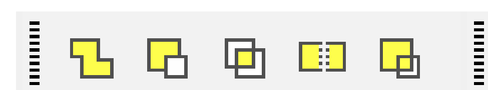
  Line, Spline, Arc, Rectangle, Ellipse, Circle, Polygon

  Draw > Line (e.g.)



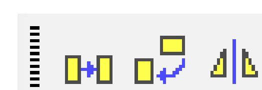
  Boolean operators

  Modeler > Boolean > Unite (e.g.)



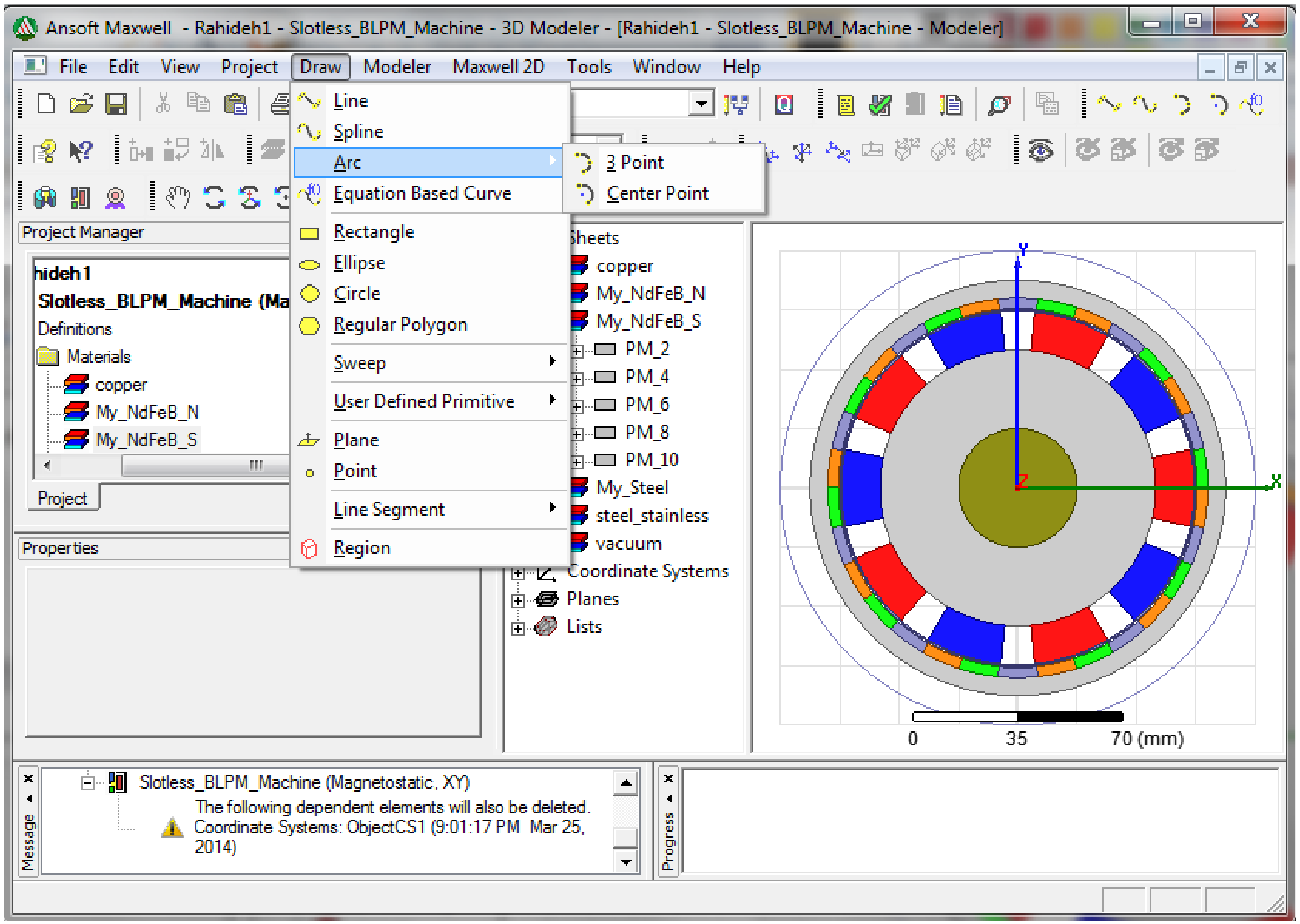
  Duplicate and Arrange

  Edit > Duplicate > Around Axis (e.g.)



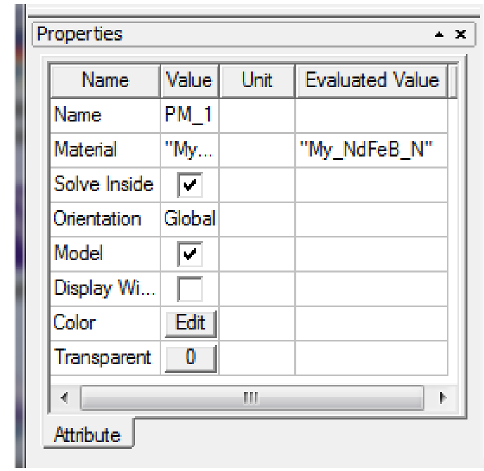
ANSYS Ansoft Maxwell®

Step 2: Define the model

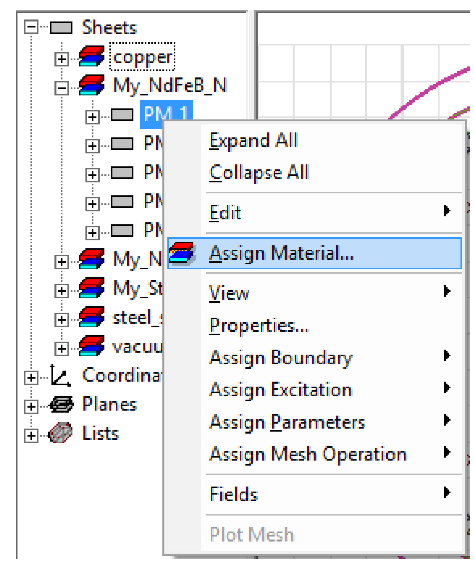


ANSYS Ansoft Maxwell®

Step 3: Assign the **material properties**



- Click on each part and from the **Properties** window either select the material from the library or define a new material.



- It can be done by right-click on the part name and select **Assign Material …**

ANSYS Ansoft Maxwell®

Step 4: Impose **boundaries**

- The boundaries can be imposed via:

  Maxwell 2D > Boundaries > Assign > Balloon (e.g.)

- Or right-click on the empty space of the Modeler window and

  Assign Boundary > Balloon (e.g.)

- The options for boundaries are:
  - **Vector Potential**: to set the magnetic vector potential, $\mathbf{A}_Z$, to a constant value on a boundary.
  - **Symmetry**: to model a fraction of a system which is repeated.
  - **Balloon**: to model the region outside the drawing space as being nearly "infinitely" large — effectively isolating the model from other sources of current or magnetic fields.
  - **Master**: to model planes of periodicity (with slave)
  - **Slave**: to model planes of periodicity (with master)

ANSYS Ansoft Maxwell®

Step 4: Impose **boundaries**

- In this example, the **balloon boundary** needs to be imposed on the outer edge of the exterior region.

- To do so first select the edge by Right-click on the modeler window and click on **Select Edges**

- Then select the outer edge of the exterior region and right-click and select *Assign Boundary > Balloon*

- And give a name (e.g. balloon1) to the balloon boundary.

ANSYS Ansoft Maxwell®

Step 5: Apply **excitation sources**

- It is possible to apply excitation source as:

  Current

  Current Density

- The excitation can be applied via:

  Maxwell 2D > Excitations > Assign > Current (e.g.)

- Or right-click on the empty space of the Modeler window and

  Assign Excitation > Current (e.g.)

ANSYS Ansoft Maxwell®

Step 6: Define **executive parameters**

- To calculate quantities such torque, force and inductances, it needs to define executive parameters.

- The parameters can be defined via:

  Maxwell 2D > Parameters > Assign > Torque (e.g.)

- Or right-click on the empty space of the Modeler window and

  Assign Parameter > Torque (e.g.)

- For force and toque calculation the rotor should be selected.
- For inductance calculation, the side-coils should be defined.

ANSYS Ansoft Maxwell®

Define **local variables**

- It is also possible to define **local variables** such as the currents of the armature windings at a single moment or as a function of time.

- To do so, use the following route

  Maxwell 2D   >   Design Properties

  and provide the name and value of each variable.

ANSYS Ansoft Maxwell®

Step 7: Select the **solution options**

- It is possible to compromise between the accuracy and complexity via solution options.
- To set up a solution use the route below:

Maxwell 2D > Analysis Set up > Add Solution Set up

and define the required solution

- The quality of mesh has a significant effect on both accuracy and complexity.
- It is possible to refine the initial mesh proposed by the software.

ANSYS Ansoft Maxwell®

Step 7: Refine **mesh**

- Mesh refinement can be done:
  - **On Selection**
    - Length Based
    - Skin Depth Based
  - **Inside Selection**
    - Length Based
- To refine mesh, the following steps should be performed:
  - Set up the restriction on the desired mesh
  - Reassign the defined restriction on the geometry
  - Apply mesh operation
  - Plot the mesh

ANSYS Ansoft Maxwell®

Step 7: How to Refine **mesh**

- To refine mesh, the following steps should be performed:
  - Set up the restriction on the desired mesh; select the area of interest and, e.g.,

    Maxwell 2D > Mesh Operations > Assign > Inside Selection > Length Based
  - Reassign the defined restriction on the geometry

    Maxwell 2D > Mesh Operations > Reassign
  - Apply mesh operation

    Maxwell 2D > Analysis Set up > Apply Mesh Operations
  - Plot the mesh

    Maxwell 2D > Fields > Plot Mesh

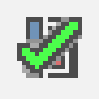
The above operations can be done by right-click on Modeler window.

ANSYS Ansoft Maxwell®

Step 8: **Solve** the problem

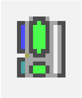
- Before solution, the problem can be checked via:

  Maxwell 2D > Validation Check

  or by clicking on the shortcut button 

- To analyze (solve) the problem

  Maxwell 2D > Analyze All

  or by clicking on the shortcut button 

ANSYS Ansoft Maxwell®

Step 9: Postprocessing

- By using the following route, the field results can be observed:

  Maxwell 2D > Fields > Fields > A > Flux Lines (e.g.)

- The field quantities to be observed consist of
  - Vector magnetic potential (Flux lines or its vector)
  - Magnetic field intensity (Magnitude or its vector)
  - Magnetic flux density (Magnitude or its vector)
  - Current density (Magnitude or its vector)
  - Energy, Co-energy and Apparent Energy
  - Ohmic losses
  - Force density on an edge or on a surface
  - Temperature

ANSYS Ansoft Maxwell®

Step 9: Postprocessing

- By using the following route, the calculated executive parameters are shown:

  Maxwell 2D > Results > Solution Data

- The executive parameters consist of
  - Force (x- and y-direction and its magnitude)
  - Torque
  - Inductance matrix

ANSYS Ansoft Maxwell®

Step 9: Postprocessing

- Plot 1D graphs, e.g. radial component of flux density vs. spatial angle.
- First a path should be defined as desired, e.g. in the middle of airgap. To do so, use “center point arc” and give it a proper name (airgap_path).
- Right-click on the airgap_path and select Fields > B > Mag_B.
- Click on the Field Calculator and define Br.
- In the project Manager right-click on Results and select Create Fields Report > Rectangular Plot.
- Set the Geometry to airgap_path select x and y to the defined parameters.

ANSYS Ansoft Maxwell®

Step 9: Postprocessing

How to plot the x- and y-components of flux density:

- From Maxwell2D→Fields→Calculator
- Select "B-vector" from "Named expressions" list
- Click on "Copy to stack"
- Find the x-component of B-vector from "vector" list → Scal? → ScalarX
- Click on "Add" and give it a name, e.g. Bx
- Repeat it for y-component of B-vector
- It is possible to have X from "Input" list→ Function and select X and click on "Add" and give it a name, e.g. Xp
- Repeat for Y.