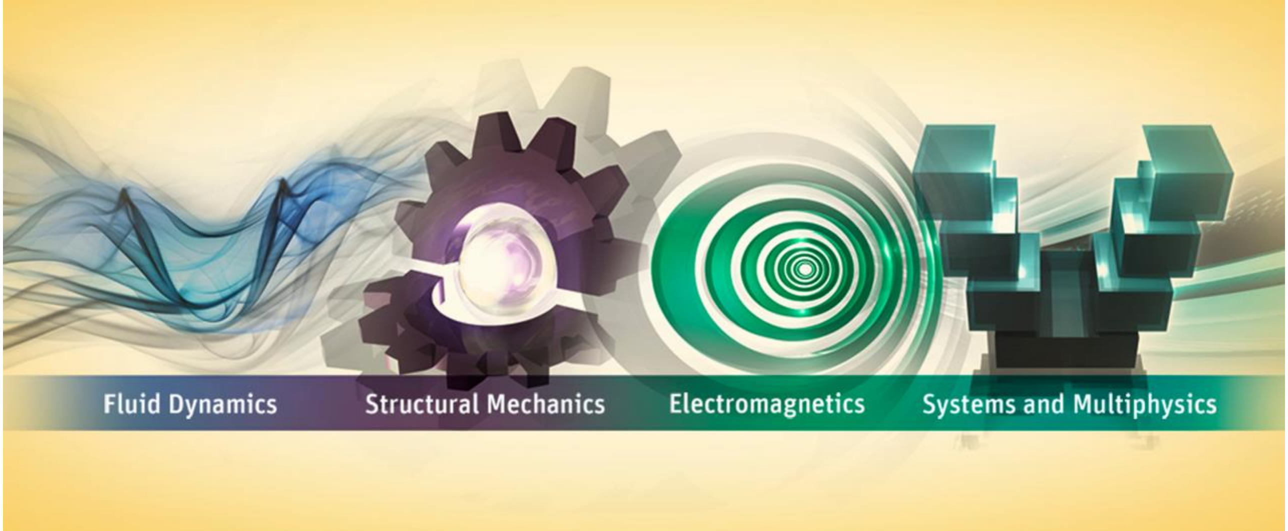


Electronic Transformer Kit Documentation

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

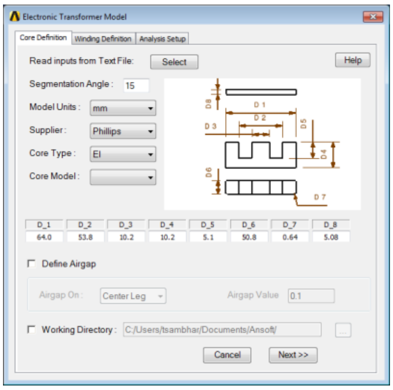
Table of Contents

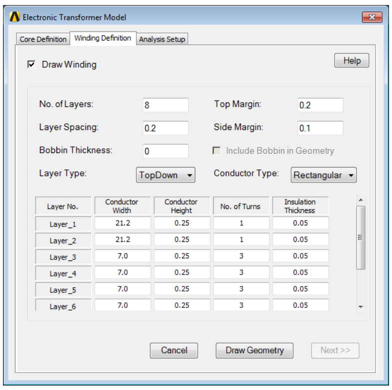
Overview of Electronic Transformer Kit

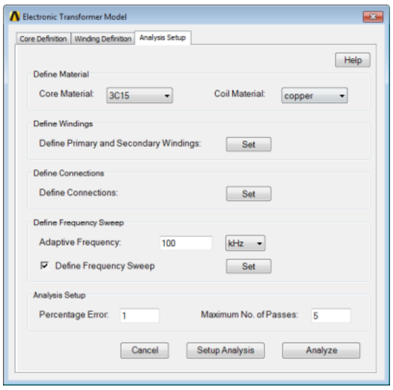
The Electronic Transformer Kit is a customized solution for electronic transformers. These devices have a ferrite core (not laminated steel), typically operate in the 100kHz frequency range (with harmonics to low MHz) and have fewer than 200 turns total in the device. The toolkit was built using Iron Python scripting and provides an easy-to-use interface to draw the geometry and setup a solution for a transformer or inductor. A database of basic topologies and materials for the commonly used cores is included in the toolkit which allows users to choose the required shape and size of the core. In addition users can define their own winding strategy using Winding definition panel (top down or concentric) which enables automatic creation of all winding turns (rectangular or circular).

The toolkit allows users to automatically setup an Eddy Current Solution with (or without) a Frequency Sweep Definition. The toolkit considers the frequency dependent core permeability and core loss Steinmetz coefficients. The toolkit also allows defining matrix connections (series or parallel) if required. Final setup can be completed through the script or outside the script. This solution process creates a frequency dependent R/L model which can be imported into ANSYS Simplorer as a Maxwell Dynamic Eddy Current component.

The Transformer Modeling script includes three input panels which must be filled out sequentially: Core Definition, Winding Definition and Analysis Setup.

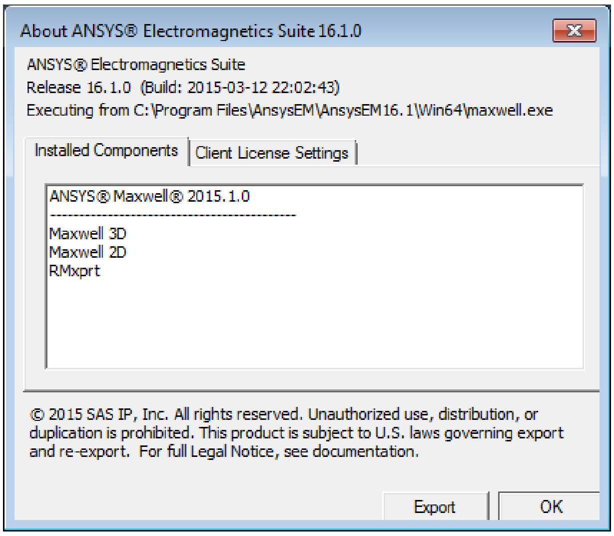






Supported Versions

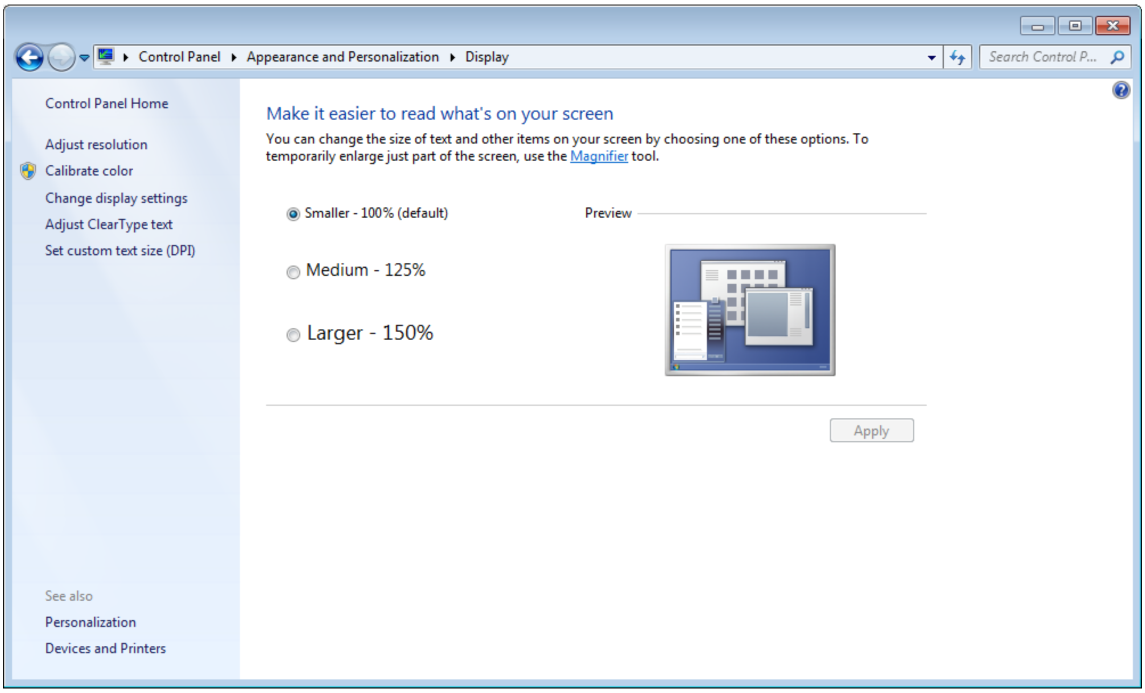
The toolkit is built for ANSYS Electromagnetic Suite 16.1 (Maxwell 2015.1). Please use the toolkit with this or any version higher than this.



Display Settings

The GUI of the Toolkit is built for Default font settings of the Windows operating system. If you have problem in viewing the GUI correctly, please ensure the following Setting

Goto ***Control Panel\Appearance and Personalization\Display*** and check if the Font setting is set to **Smaller.** If this setting is set to Medium or Larger, please change it to Smaller to view the GUI correctly.



Overview of three required Input Panels

Panel #1 – Core Definition

Reading Inputs from text

If a model was created previously, a .tab data file can be read in to recreate the same model. Otherwise, the user can skip this button.

All data required by the three input panels (core definition, winding definition, and analysis setup) is automatically saved in a .tab delimited file in the same directory as the project file with every execution of the script. Once this file is available, users can click on the “Select” button adjacent to “Read Inputs from Text File” and browse to the tab file location to it. After selecting, all previous inputs will be automatically added the panels.

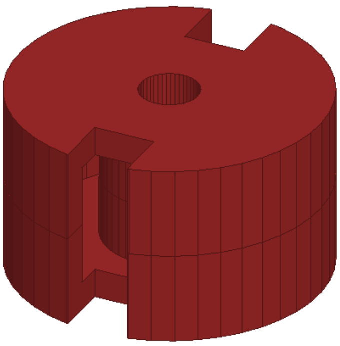
The name of the .tab file is appended with Project name and design name. This .tab file can be used for modification or recreating the design in subsequent analysis

Sample of .tab files are available in the script folder under “\CoreUDM\Demo_Planar.tab” and “\CoreUDM\Demo_WireWound.tab”

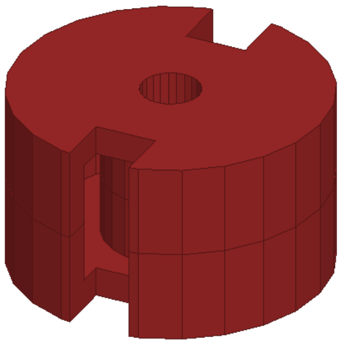
Segmentation Angle

The angle defined in the textbox will be used as the default segmentation angle for the geometry being created using the Toolkit. This value is used to segment major curvatures on core and winding. However fillets will not be segmented.

The default value is set to 15 degrees around 360 degree surface. The value of the segmentation angle must be 0 < angle < 20 degrees to avoid poor geometry representation.



Segmentation Angle: 8 deg



Segmentation Angle: 15 deg

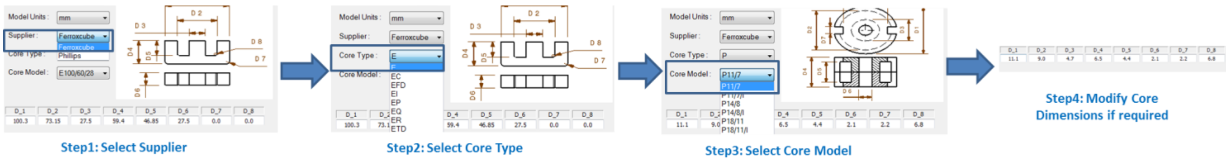
Model Units

The toolkit allows definition of geometry in either “mm” or “inches”. The geometry of core and winding will be created using the selected units.

If the units are changed, all parameters entered in the UI will be scaled to new units. The displayed values will be rounded to four decimal places. However the values used in actual geometry creation will be with much higher precision.

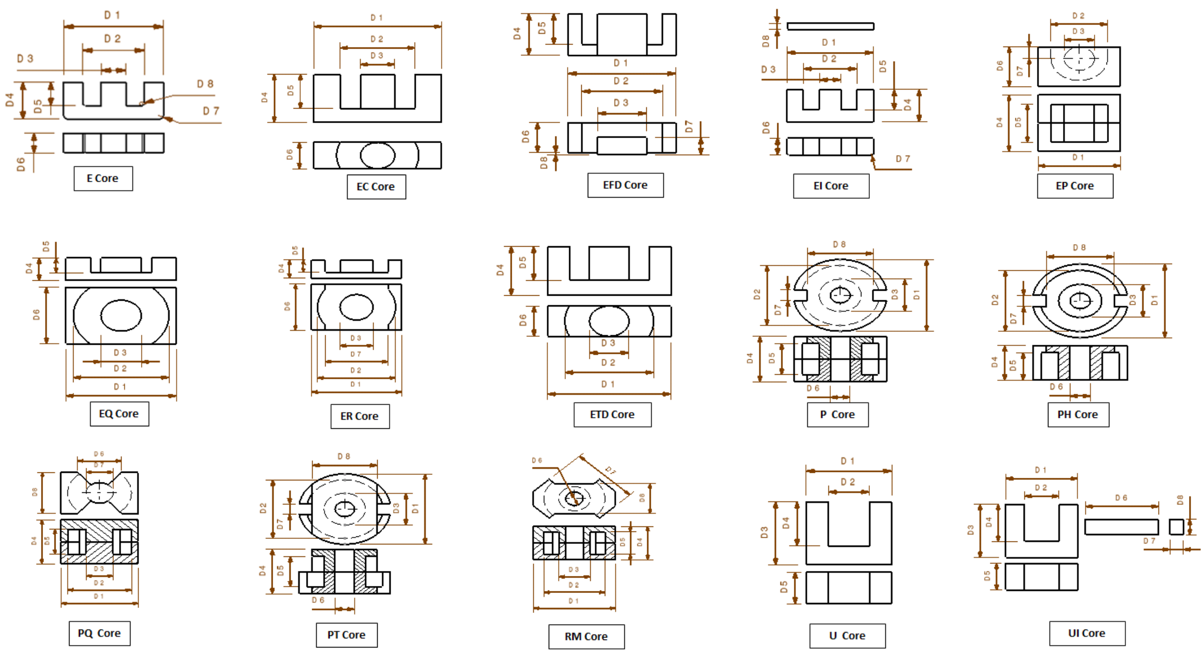
Core Parameter Definition

The toolkit contains a database of cores from several suppliers. Users can select the core supplier and then select the core type from available 15 topologies. For each core topology, different core models are provided for each supplier. The dimensions of the selected core model will be listed in the table, as shown below. Users can accept the core dimensions as displayed in the panel or manually modify them as per their requirements.



Core Types

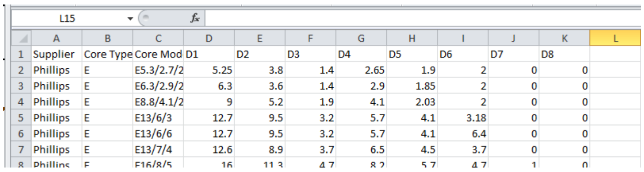
There are 15 core topologies currently supported by the toolkit: E, EC, EFD, EI, EP, EQ, ER, ETD, P, PH, PQ, PT, RM, U, UI. These basic topologies are used frequently and are supplied by most of the manufacturers. The description of the dimensions of all these core topologies is given in below images.



Adding Core Models

Core models from “Phillips” and “Ferroxcube” datasheets have been included in the database. However, additional core models from other suppliers can be manually added by the user. In order to add a new core model to the database, users can open the “CoreData.tab” file using Excel or a text editor and add the Supplier, Core Type and the appropriate core dimensions for that shape. Once edited, users can save the file again as tab delimited file overwriting the existing file. This will add the defined material to database and will be available through the input panels.

Note that core topologies are limited to the 15 options listed above so that any new core model should correspond to one of the above core shapes. It is not possible to add a custom core topology.

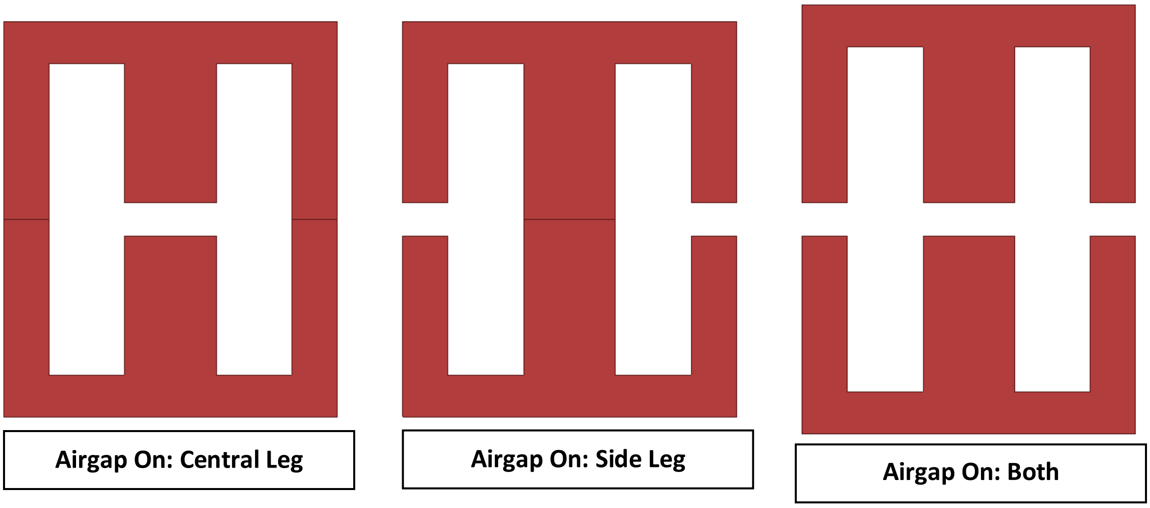


	A	B	C	D	E	F	G	H	I	J	K	L
1	Supplier	Core Type	Core Mod	D1	D2	D3	D4	D5	D6	D7	D8	
2	Phillips	E	E5.3/2.7/2	5.25	3.8	1.4	2.65	1.9	2	0	0	
3	Phillips	E	E6.3/2.9/2	6.3	3.6	1.4	2.9	1.85	2	0	0	
4	Phillips	E	E8.8/4.1/2	9	5.2	1.9	4.1	2.03	2	0	0	
5	Phillips	E	E13/6/3	12.7	9.5	3.2	5.7	4.1	3.18	0	0	
6	Phillips	E	E13/6/6	12.7	9.5	3.2	5.7	4.1	6.4	0	0	
7	Phillips	E	E13/7/4	12.6	8.9	3.7	6.5	4.5	3.7	0	0	
8	Phillips	E	E16/8/5	16	11.3	4.7	8.2	5.7	4.7	1	0	

Defining Airgap

In order to model an airgap in the core, user needs to select the "Define Airgap" checkbox. If this checkbox is checked, user will have option to define airgap on central leg or side legs or both. The user can enter the size of the airgap to be modelled and the specified airgap will be included in core geometry

Note: When an airgap is defined on the "Central Leg" or "Side Legs", the overall dimensions of core remain unchanged (the airgap is created by removing material from the core at the specified leg.) However when airgap is defined as "Both", the core halves are moved apart to add airgaps instead of removing material from core. Thus overall height of the core will increase.



Defining Working Directory

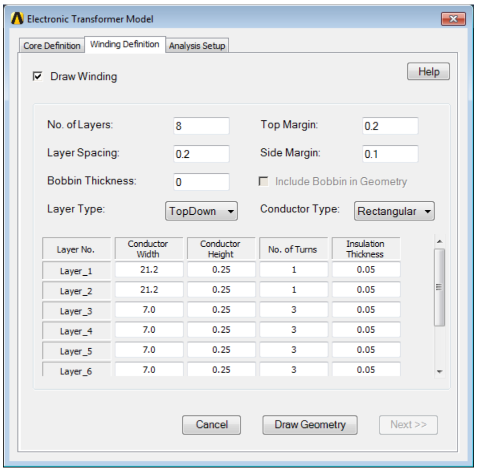
The Working directory definition allows users to define location where the file will be saved. User should have permission to write in the specified directory.

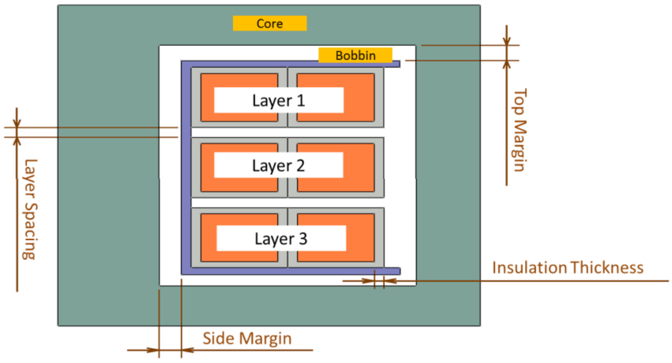
By default the directory is pointed towards the Maxwell Project directory (Tools > Options > General Options) or the directory of the project in which the toolkit is being executed. Toolkit users can change the working directory by enabling the checkbox to define the working directory. Users can either directly enter the path in the textbox or use the button adjacent to it and browse to the required folder.

When all inputs on the core definition panel are completed, the user can click on **"Next>>"** or alternatively click on the **"Winding Definition"** tab at the top of the panel to proceed to the 2nd panel. Clicking on **"Cancel"** will cancel and close the script.

Panel #2 - Winding Definition

Parameters related to winding specification are defined under the Winding Definition tab.





The meaning and specification of all the terms under this tab are described below.

Top and Side Margins

Top and side margins are the spacing between core and the Bobbin. Top margins are spacing at top and bottom of the core while side margin is the spacing along the core center leg. This value can be zero or greater than zero.

Layer Spacing

Layer spacing is the spacing between two adjacent layers of the core. This value can be zero or greater than zero.

Bobbin Thickness

Thickness of the bobbin on which winding is wound. If this value is set to zero, bobbin is not considered. If Bobbin Thickness is more than zero, the value is considered for positioning the winding even if include bobbin is unchecked.

Include Bobbin

Selecting this checkbox will model the geometry of bobbin. If unchecked, bobbin thickness value is used to determine the position of windings while bobbin is not drawn explicitly.

Layer Definition

Number of Layers

Total number of layers in the winding. Once the entry for number of layers is done, table below will be modified to facilitate entry of parameters for all layers.

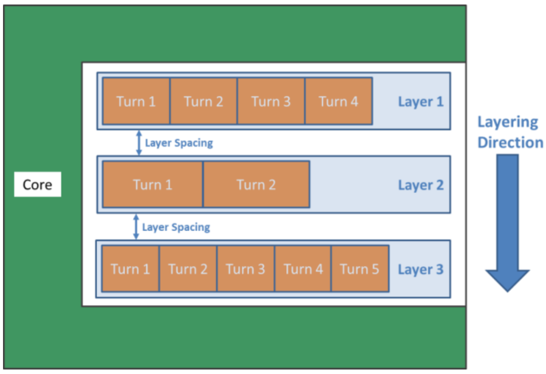
Layer Types

TopDown

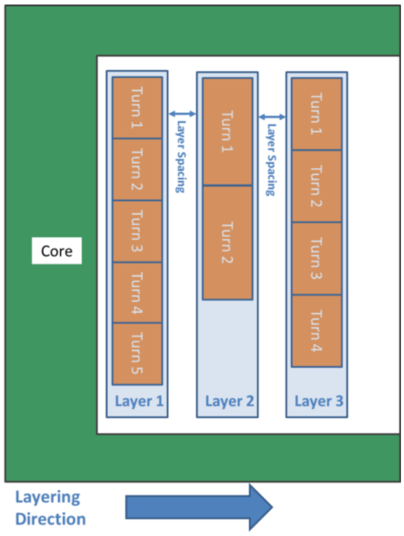
Topdown Layer arrangement indicates that layers are arranged from top to bottom in the core. Turns in each layer in this arrangement will be added radially outwards.

Concentric

Concentric Layer arrangement indicates that layers are arranged radially with layer number incremented radially outwards. Turns in each layer in this arrangement will be vertical (top to bottom).



Layer Type: TopDown

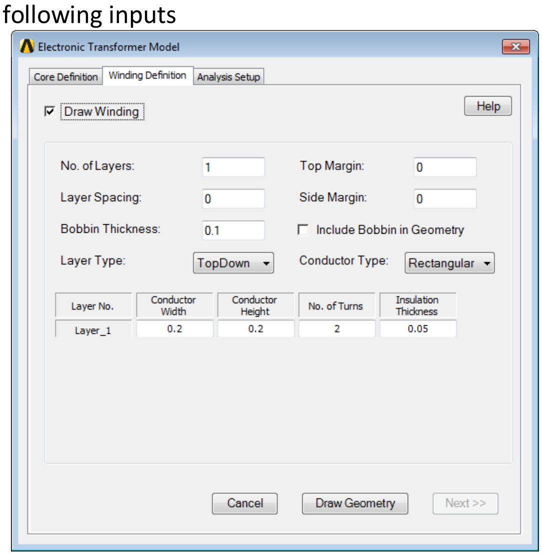


Layer Type: Concentric

Conductor Type

Rectangular

When Conductor type is set to Rectangular, it is considered that cross section of the conductor is rectangular. Rectangular conductor definition will require following inputs



Conductor Width

Width of the conductor to be specified

Conductor Height

Height of the conductor to be specified

Number of Turns

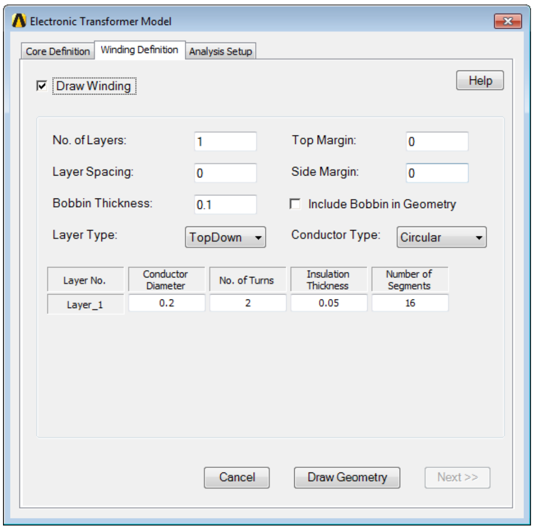
Number of conductor turns in each layer

Insulation Thickness

Thickness of the insulation on the conductor. Even though insulation is not modelled geometrically, the value specified for insulation thickness will be considered for modelling location of the turns

Circular

When Conductor type is set to Circular, it is considered that the cross section of the conductor is Circular. Circular conductor definition will require following inputs



Conductor Diameter

Diameter of the conductor to be specified

Number of Turns

Number of conductor turns in each layer

Insulation Thickness

Thickness of the insulation on the conductor. Even though insulation is not modelled geometrically, the value specified for insulation thickness will be considered for modelling location of the turns

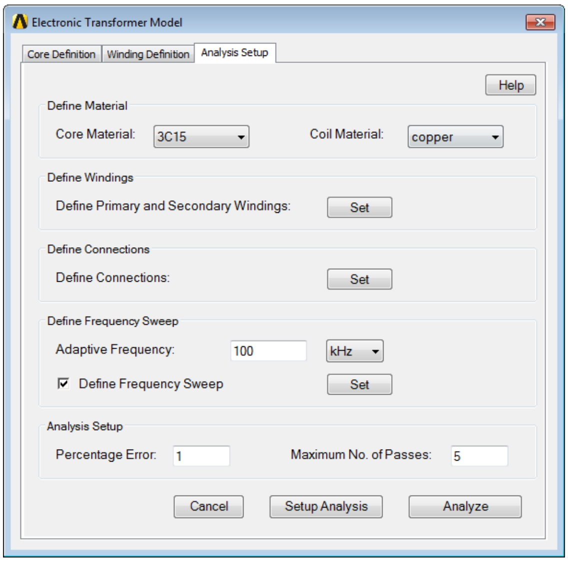
Number of Segments

The value specified in this column will be used to segment the circular conductor. The value specified should be greater than 8. Any value less than 8 is not permitted.

When all inputs on the **"Winding Definition"** panel are completed, the user can click on **"Draw Geometry"** to create the model in Maxwell. This may create a model with overlapping objects, which will need to be corrected. Clicking on **"Next>>"** or alternatively click on the "Analysis Setup" tab at the top of the panel will proceed to the 3rd panel. Clicking on **"Cancel"** will cancel and close the script.

Panel #3 - Analysis Setup

Parameters related to Analysis Setup are defined under **"Analysis Setup"** tab of the GUI. Once the geometry is drawn using **"Draw Geometry"** button on winding Definition tab, the options on Analysis Setup tab will be enabled.



Define Material

Select the ferrite core material (as listed in CoreData.tab material file) and coil material (copper or aluminum) from the pull down menus.

Adding a Core Material

In order to add a new core material, users can follow below steps

1. Create a tab file for frequency versus permeability for the required core. Sheet Scan option available in Maxwell can help for creation of the tab fil.
2. Name the tab file same as the name of the material to be added
3. Place the tab file in the folder "CoreUDM /MaterialData"
4. Open "matdata.tab" file, which is in the folder "CoreUDM /MaterialData",in Excel
5. Add a row for the material to be added and specify name and other material properties
6. Save the file with same name.

Once this is done, the material should be available for selection for next run of the Toolkit

Primary and Secondary Definition

Once all the winding layers are drawn, user needs to define which of the layers correspond to Primary winding and which layers will be considered as secondary winding. The definition of Primary and secondary will be sued to define direction of current. Current Direction in secondary is assigned as opposite to Primary.

All layers listed in the dialog should be defined as either Primary or Secondary. Further operations will be enabled only after all layers are defined as Primary or Secondary.

Define Connection

Once Primary/Secondary definition is complete, the option to define connection will be enabled. This is an optional step which will allow users to define Winding Groups. The result of this operation will be same as the Post-processing operation in Matrix Definition window. The interface allows to group the winding together and also enables users to define name of the groups and number of parallel branches in the group.

Frequency Definition

Adaptive Frequency

The Frequency defined under Adaptive Frequency is sued for solving the fields with Adaptive mesh refinement. It is advisable to define highest frequency of the frequency range to be solved under the adaptive frequency. This ensures that the skin depths are resolved for all the frequency range to be solved.

Frequency Sweep

The checkbox "Define Frequency Sweep" enables to setup a frequency range on which the results need to be computed. After the checkbox is checked, users can click on the button "Set" to define the frequency range to be solved. Users can Define Start and Stop frequency together with number of samples to be solved in that range. Further the sampling method can also be defined as Linear or Logarithmic.

Note: Frequency Sweep will be necessary if user is interested in System Simulation using State Space Model extracted from Maxwell solution.

Analysis Setup

The details mentioned under this section will be used to determine the level of accuracy required in the simulation.

Percentage Error

The Percent Error allows you to control the desired solution accuracy. Smaller values produce more accurate (but slower) solutions.

Maximum Number of Passes

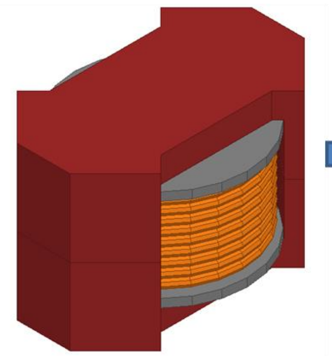
The Maximum Number of Passes is the maximum number of mesh refinement cycles you would like Maxwell to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops even if the Error criterion is not achieved.

When all inputs on the **"Analysis"** panel are completed, the user can click on "Setup Analysis" to setup the model in Maxwell and close the script. Clicking on **"Analyze"** will solve Maxwell and close the script. Clicking on **"Cancel"** will cancel and close the script.

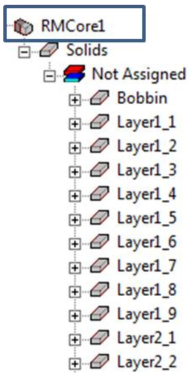
Maxwell Project Variables

All dimensions are saved as parameters on the Properties tab for the component model. These can be manually changed as desired (instead of re-running Python script) or parameterized for Parametric/Optimetrics analysis.

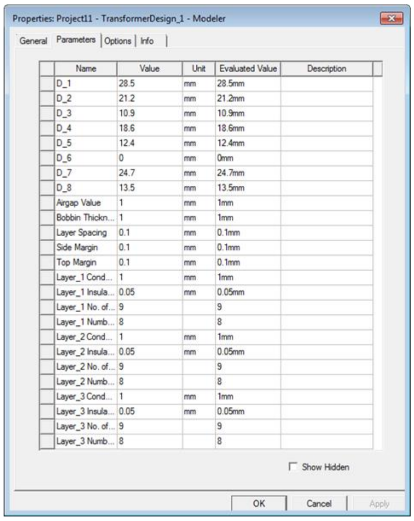
Note that while dimensions can be modified, the script needs to be re-run in order to change the number of layers or number of turns.



Draw Geometry using Toolkit



Double click on the created UDM



Name	Value	Unit	Evaluated Value	Description
D_1	28.5	mm	28.5mm	
D_2	21.2	mm	21.2mm	
D_3	10.9	mm	10.9mm	
D_4	18.6	mm	18.6mm	
D_5	12.4	mm	12.4mm	
D_6	0	mm	0mm	
D_7	24.7	mm	24.7mm	
D_8	13.5	mm	13.5mm	
Airgap Value	1	mm	1mm	
Bobbin Thickn...	1	mm	1mm	
Layer Spacing	0.1	mm	0.1mm	
Side Margin	0.1	mm	0.1mm	
Top Margin	0.1	mm	0.1mm	
Layer_1 Cond...	1	mm	1mm	
Layer_1 Insula...	0.05	mm	0.05mm	
Layer_1 No. of...	9		9	
Layer_1 Numb...	8		8	
Layer_2 Cond...	1	mm	1mm	
Layer_2 Insula...	0.05	mm	0.05mm	
Layer_2 No. of...	9		9	
Layer_2 Numb...	8		8	
Layer_3 Cond...	1	mm	1mm	
Layer_3 Insula...	0.05	mm	0.05mm	
Layer_3 No. of...	9		9	
Layer_3 Numb...	8		8	

Geometrical Parameters for changing geometry