Dr. Mühlhaus Consulting & Software GmbH

www.muehlhaus.com
info@muehlhaus.com

Tutorial: Getting Started with Inductor Toolkit PCB

Table of contents:

Document revised: 27. April 2015
Document revision: 1.0

Introduction

Mühlhaus Inductor Toolkit PCB assists you with the design of "optimum" inductors in PCB technologies.

It very quickly makes you an inductor design expert, using a very efficient synthesis approach with EM-accurate results. You only need to know the electrical properties that you want, and Inductor Toolkit will find the best possible layout for you. As you are getting more experienced in this, you can later customize the tool and settings to your preferences.

This document will provide a quick start with several examples, starting from the most basic geometries.

Installation

Before you can use Mühlhaus RFIC InductorToolkit, you have to install it to your favourite designkits. The procedure for that is similar to other ADS PDKs: in the ADS main windows, go to *DesignKits > Unzip Design Kit ...* and install the PDK.

Installation path

You must install Inductor Toolkit with <u>write access</u> for the users at file level, because users can customize settings and schematics. For example, if you store your ADS workspaces to $HOME/ads2012.08, you could create a new directory $HOME/designkits and unzip Inductor Toolkit to that location. **Do not install to a shared server directory where users have read access only!**

Adding Inductor Toolkit to an ADS workspace

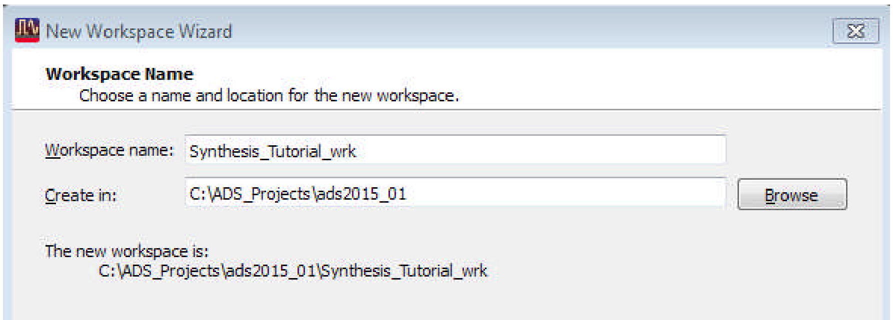
Mühlhaus Inductor Toolkit PCB is implemented as an ADS design kit.

Unlike other PDKs, **the inductor toolkit libraries must be writable**, because emModels and schematics from Inductor_EM_Models_lib are updated during synthesis.

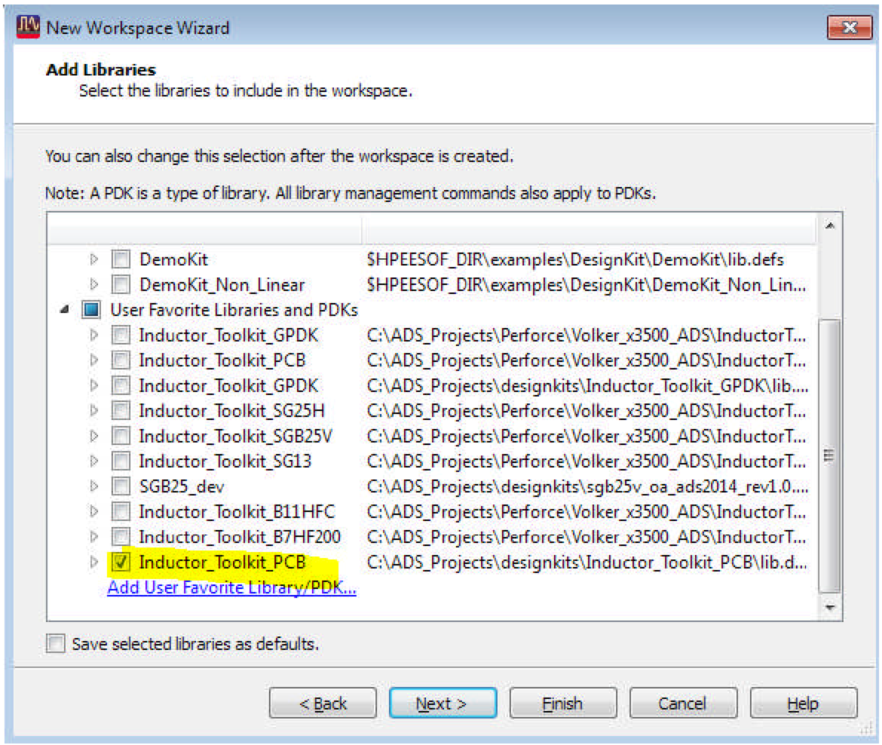
Create a new example workspace

We want to use a new workspace for this tutorial. Start ADS and from the menu, do File > New Workspace.

Follow the wizard, and set the workspace name to Synthesis_Tutorial_wrk

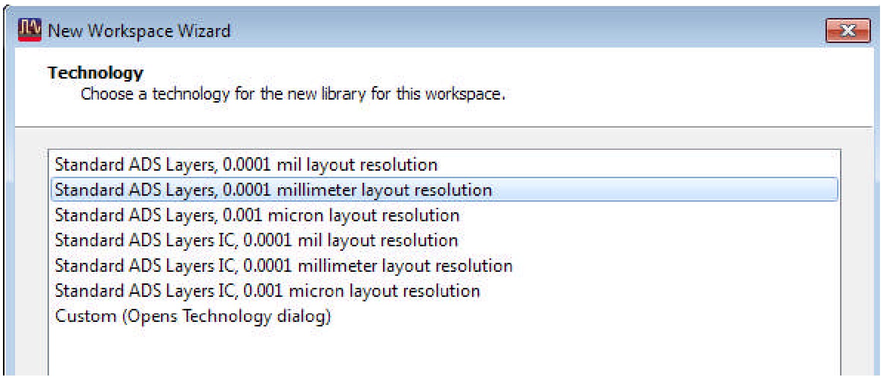


Next, you need to set the libraries/PDKs to use with the workspace. Select the Analog/RF libraries **and** the Inductor_Toolkit_PCB designkit.



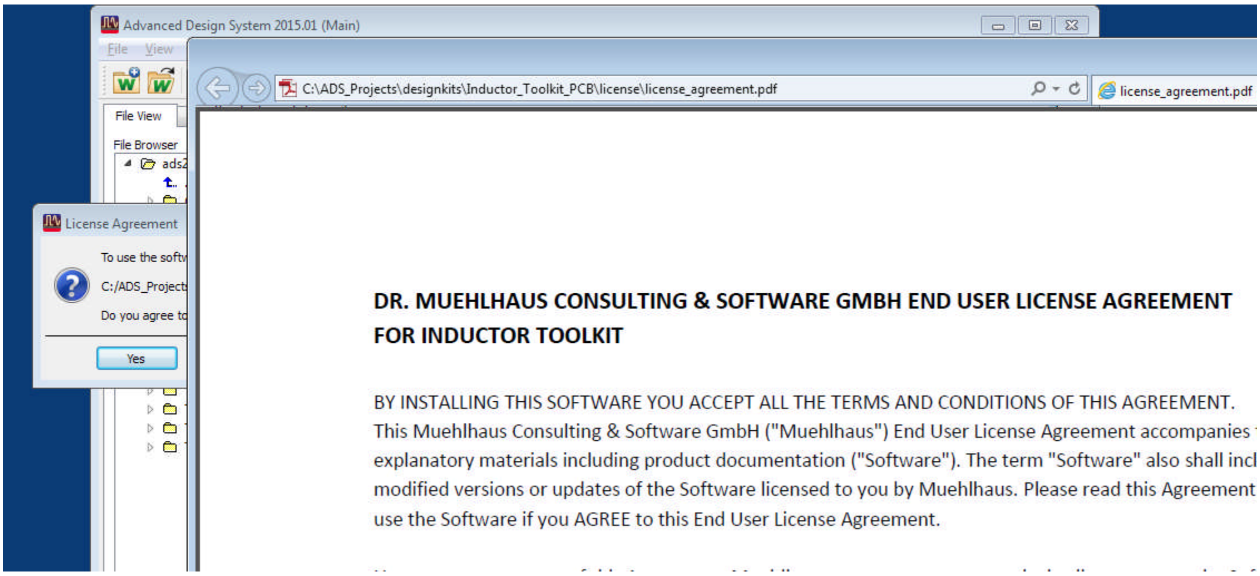
For the library name, accept the default name (derived from the workspace name).

Next, you have to specify the technology used in your project. Select the standard ADS layers with unit millimeter.



Press the Finish button to complete the workspace setup.

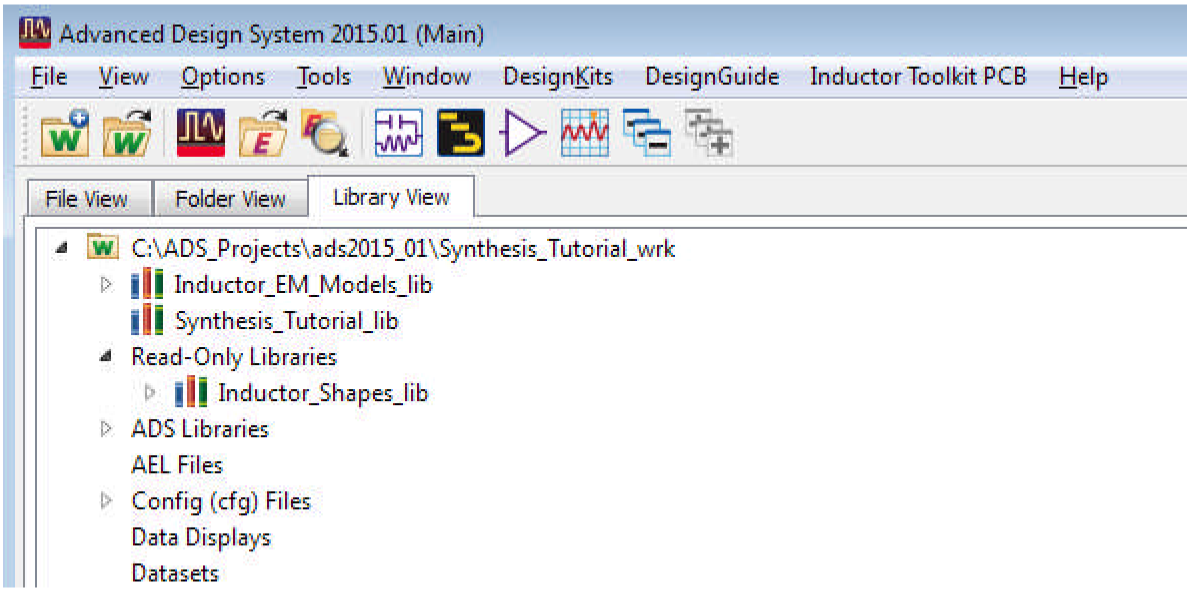
When Inductor Toolkit is added to an ADS workspace for the first time, a license agreement dialog will be shown. To use the software, **carefully read the agreement** and **press "Yes" if you agree to the license terms and conditions**. If you do not agree, you may not use the software and need to contact Dr. Mühlhaus Consulting & Software GmbH.



Your workspace is now ready for use, and you should see "Inductor Toolkit PCB" in the ADS main window's menu.

For a better understanding of the library and PDK configuration, we recommend to set the ADS main window to "Library View". Just click on the corresponding tab at the top of the project tree display.

You can now see the structure of the libraries in your workspace:



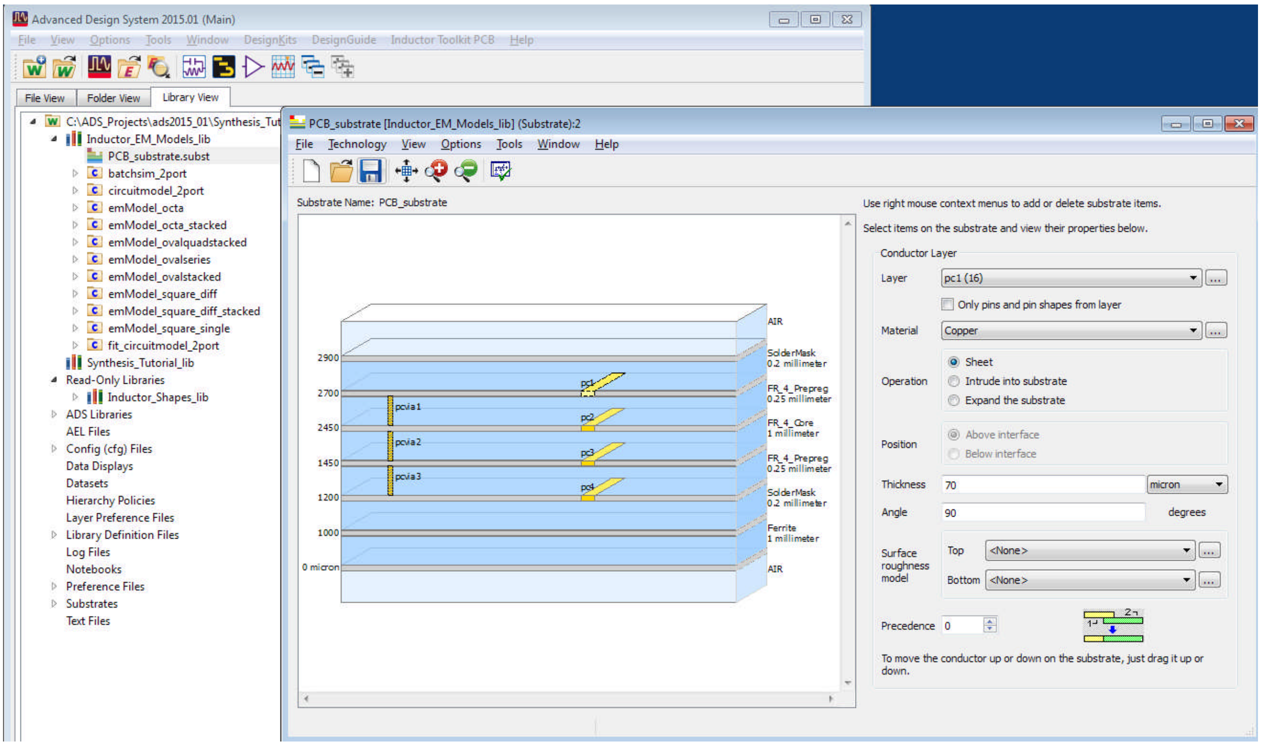
The "Synthesis_Tutorial_lib" was created with the new workspace, and is empty at this point. This is where we will store new inductors synthesized in this tutorial.

"Inductor_EM_Models_lib" is part of Inductor Toolkit, and contains the pre-defined inductor models and schematics for inductor synthesis. These will be used automatically "behind the scenes" when you synthesize new inductors.

In addition, there are read-only libraries for the foundry PDK, and the Inductor_Shapes_lib library with the inductor layout definitions.

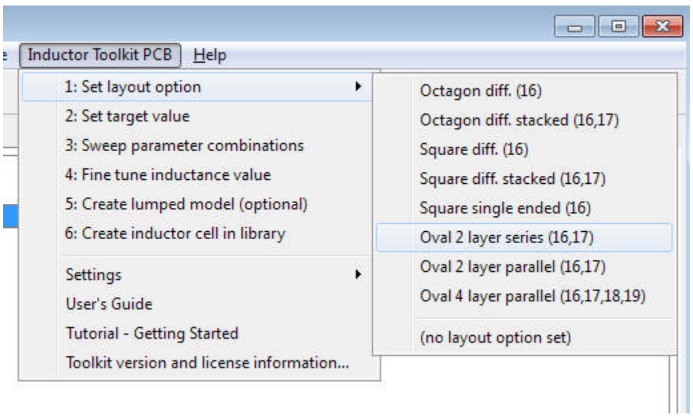
Design your first inductor

The examples below use the pre-configured 4 layer PCB technology. With other technologies, the resulting Q factors and optimum geometries will be different, but don't worry: that is what Inductor Toolkit is about, to find the optimum values for your specs and your technology.



Let's start and design your first inductor: 12µH for target frequency 100kHz.

Synthesis steps are numbered. No surprise, we start with step **1: Set layout option**. In the menu, click on "**Oval 2 layer series (16,17)**." This is an oval inductor with the default aspect ratio, with conductor on two layers connected in series for increased inductance. Layer numbers (16,17) tell you on which ADS layers the inductor coil is drawn.

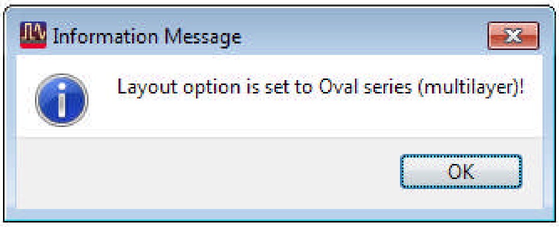


For the oval shapes, the are different versions:

- **Two** metal layers with the conductors connected in **series**,
- **Two** metal layers with the conductors connected in **parallel** and
- **Four** metal layers with the conductors connected in **parallel**.

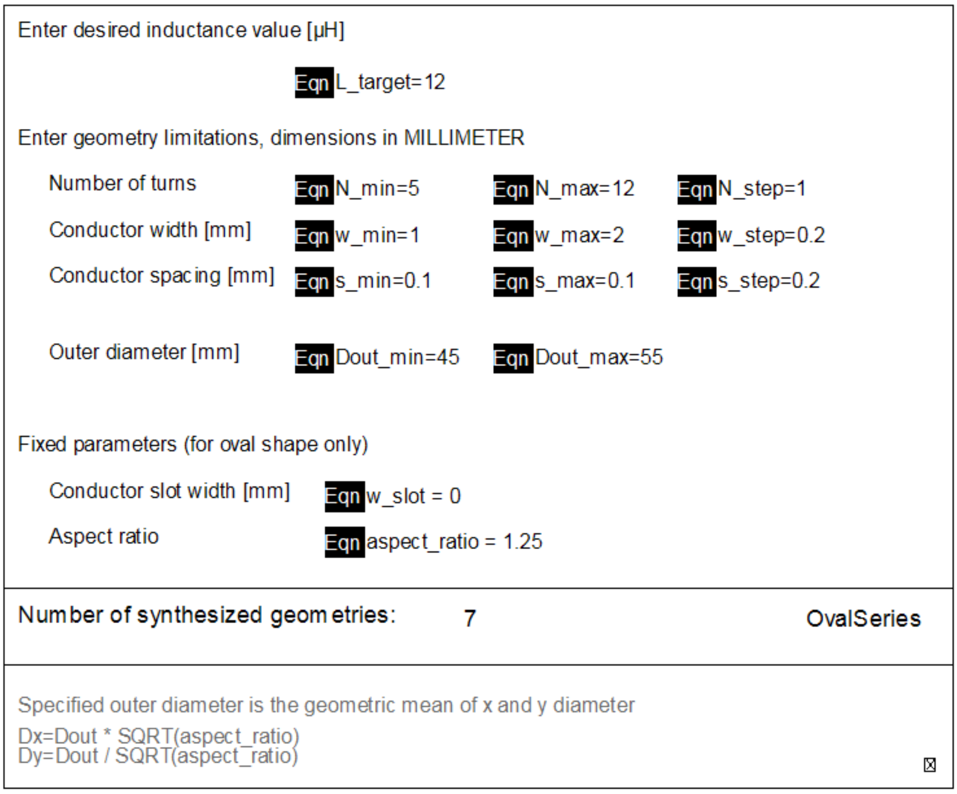
In addition to the oval shapes designed for PCB use, there are generic symmetric shapes (octagon and square) and a non-symmetric square inductor.

In this tutorial, we just go ahead with the "**Oval 2 layer series (16,17).**""



Now that we have set the layout option, go to step **2: Set target value.** This opens a data display form where the target value and geometry limitations are specified.

Change the target value to 12 (for 12µH) and set the geometry limits as shown below:



This defines the geometry space where Inductor Toolkit will search for the optimum inductor: number of turns from 5 to 12 turns in full turns, conductor width from 1 to 2 mm with 0.2mm step size, spacing between turns fixed to 0.1mm, and inductor outer diameter between 45mm and 55mm. That outer diameter is the geometric mean of the x and y size, and the actual shape can be set with the Aspect Ratio parameter.

Also, for the oval shapes a slot width can be defined. For non-zero slot width, a slot will be created in the center of the conductor trace, with the given slot width. If invalid parameter combinations are detected, resulting in traces or slots that are too small, no slot will be created.

> **For information only:** Geometries are changed against the technology limits defined in *<Inductor-Toolkit-Path>/de/ael/technology.ael*

technology.ael - Editor

Datei Bearbeiten Format Ansicht ?

```
// ========================== Metal line dimension rules =============

  // dimension in micron
  decl min_linewidth = 100;    // layout rule for minimum line width
  decl max_linewidth = 5000;   // layout rule for maximum line width
  decl min_spacing   = 50;     // layout rule for minimum slot width
```

> This helps to generate only valid layouts that can be manufactured. You can adjust the dimensions rules in the technology file as needed, to match your actual manufacturing limitations.

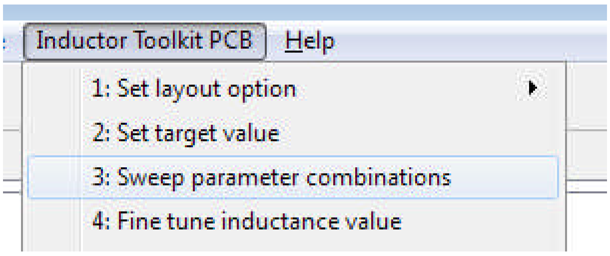
Back to our example:
The "Number of synthesized geometeries" at the bottom shows how many different layouts are possible with these geometry limitations, for the specified target value. That value is calculated in real time, based on your input.

You might wonder why we don't sweep the spacing between the turns. The reason is that Q factor is often not sensitive to that parameter, and it would be inefficient to always sweep it during our inductor optimization. We usually keep it at some fixed value, but you could sweep across a few values if you wish..

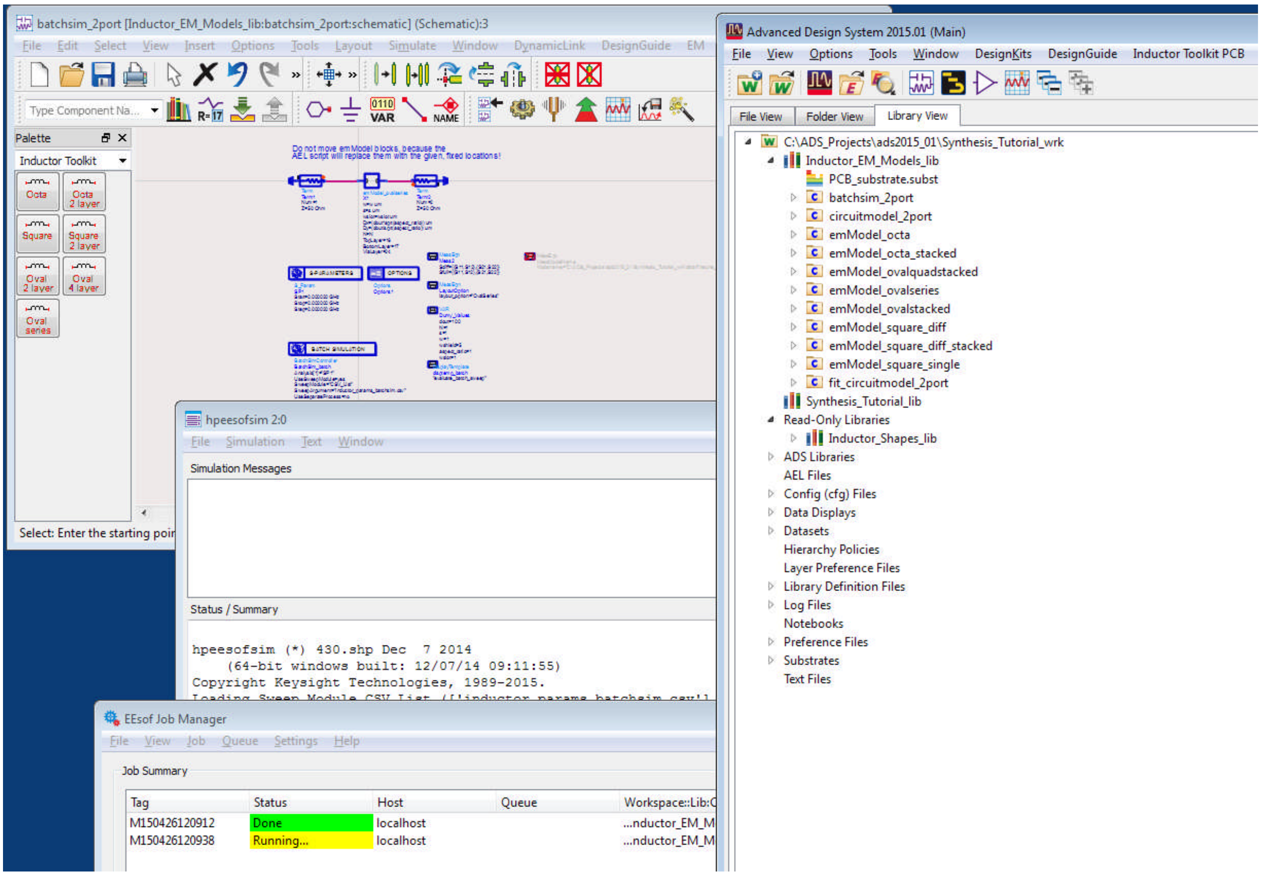
This covers the geometry space that we have specified, we find 7 different layouts which results in the desired 12µH inductor. We would fine more possible inductors if we allow a wider range of diameters

Save the data display, so that we can proceed to the next step.

Inductor Toolkit has calculated the geometry parameters for 7 different inductors that match our specification. We are now ready to EM simulate all of them, to find the inductor with the best possible Q factor. In the Inductor Toolkit menu, click on **3: Sweep parameter combinations**.

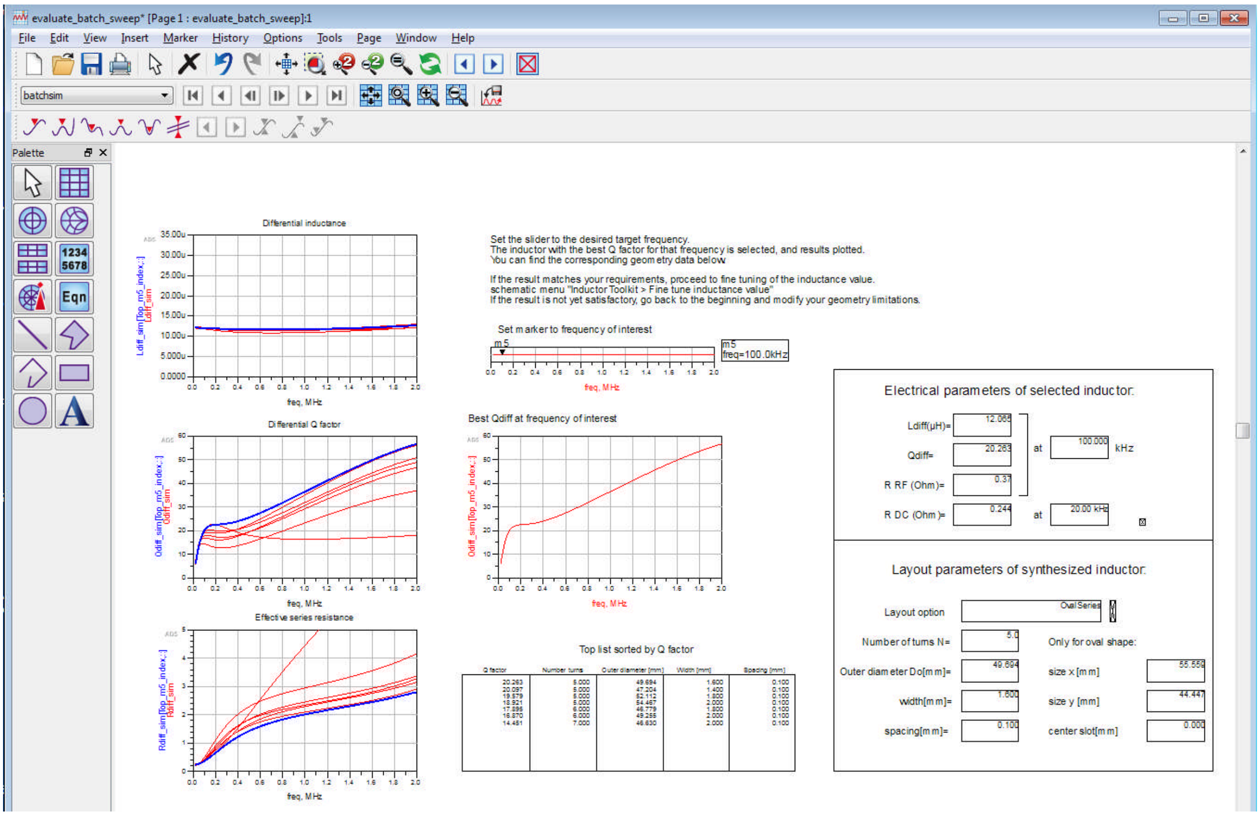


This will bring up a schematic window (batchsim_2port) with Momentum emModels, set up for batch simulation of the previously calculated geometries. The simulation of that schematic is started automatically

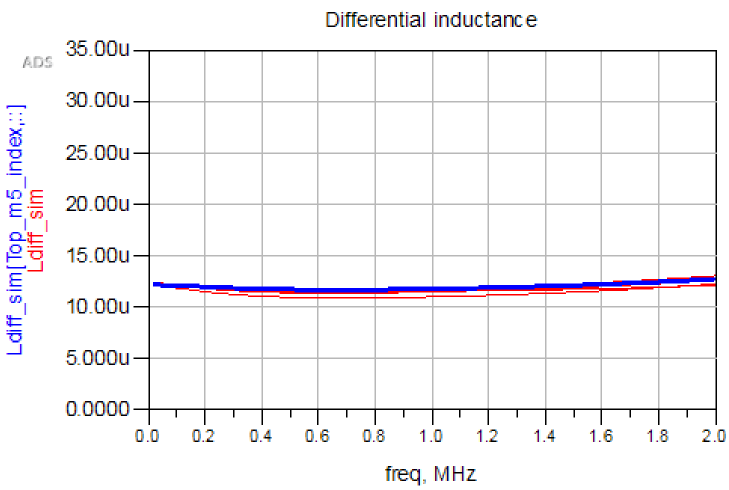


It takes a few minutes to finish this simulation, because all the layouts are now passed to Momentum for EM simulation. Then, you will see a data display where results are shown and compared in terms in inductor parameters L and Q.

Here is the overview of the result page, which is automatically shown after simulation. The details are discussed and explained below.



On the top left side, the effective inductance is plotted for the different layouts. This is the differential inductance, as measured differentially between the inductor pins.



All the inductors have a low frequency inductance that is close to the specified 12µH target value. This confirms that the synthesis of geometries in step 2 worked well.

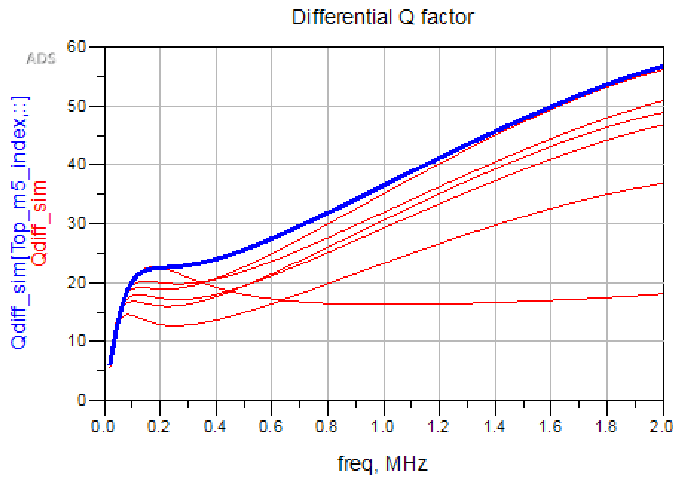
(When a ferrite is included, it also means that the manual scale factor for inductance with ferrite works well).

#
At high frequency, parasitic capacitance can cause a change in effective inductance, but that is not an issue at the low MHz frequencies that we have simulated here.

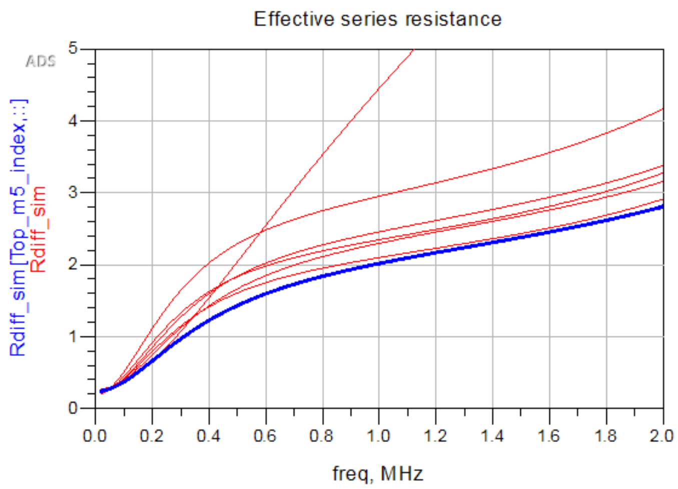
Let's look at the Q factor, because that is what we want to optimize.

The maximum Q factor is very different between the layouts, as expected.

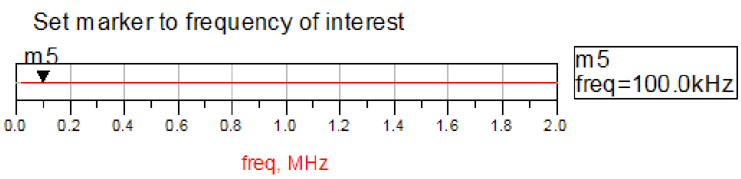
One of the Q factor curves is highlighted in blue color. That is the result with the highest Q factor at the frequency of interest, set with the slider on the top right side.



Also, the effective series resistance (including skin effect, proximity effects etc.) is shown. That changes with frequency, and some layouts might be better in one frequency range but worse at other frequencies.

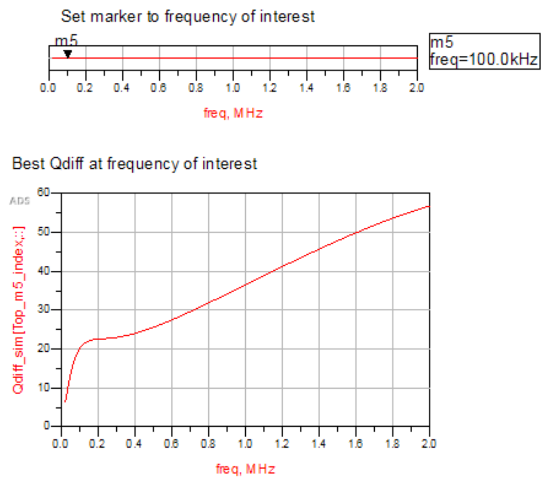


You can change the frequency of interest by moving the marker (hint: to select the marker, click on the readout box right from the slider), or by typing the value into the readout box. Notice how the values for the best inductor change as you change the target frequency.



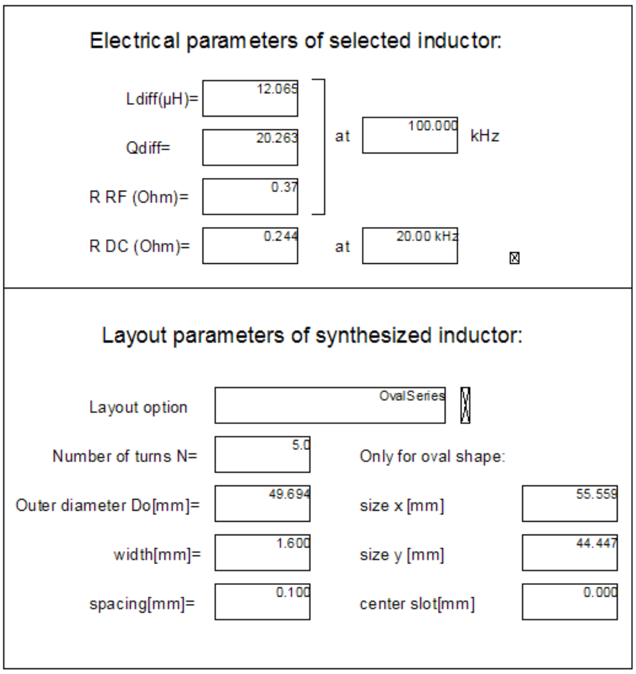
When the frequency is changed to our goal of 100kHz, the highlighted “best” inductor result is still the same. This is because for this geometry and layer configuration, the same layout is best at all simulated frequencies.

The best inductor's parameters are shown on the top right side:



Top list sorted by Q factor

Q factor	Number turns	Outer diameter [mm]	Width [mm]	Spacing [mm]
20.263	5.000	49.694	1.600	0.100
20.097	5.000	47.204	1.400	0.100
19.579	5.000	52.112	1.800	0.100
18.921	5.000	54.467	2.000	0.100
17.895	6.000	46.779	1.800	0.100
16.870	6.000	49.255	2.000	0.100
14.451	7.000	46.630	2.000	0.100

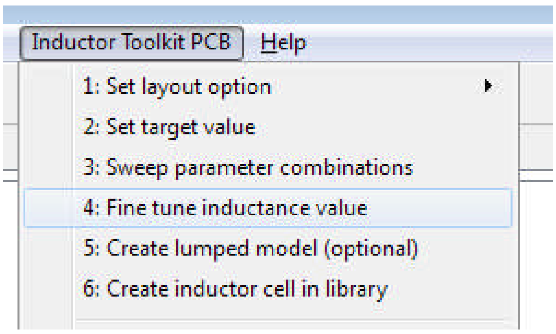


This is the best inductor within this batch simulation, and will be used to derive the best possible inductor layout:

- N=5 turns (range was set to N=5...12 turns)
- Diameter Do = 49.7mm (actually 44.4mm x 55.6mm, with aspect ratio 1.25)
- Line width w= 1.6mm (range was set to 1mm ... 2mm in steps of 0.2mm)
- Spacing s= 0.1mm (fixed)
- No slot

If we are satisfied with this inductor's performance, we can now proceed to the next step. Otherwise, we have to modify the geometry space, and allow inductors that are physically larger. We will look at that later, in the next tutorial.

So now, we have finished the batch simulation step, and selected the best performing inductor from the list, for our frequency of interest. The next step is **4: Fine tune inductance value**.

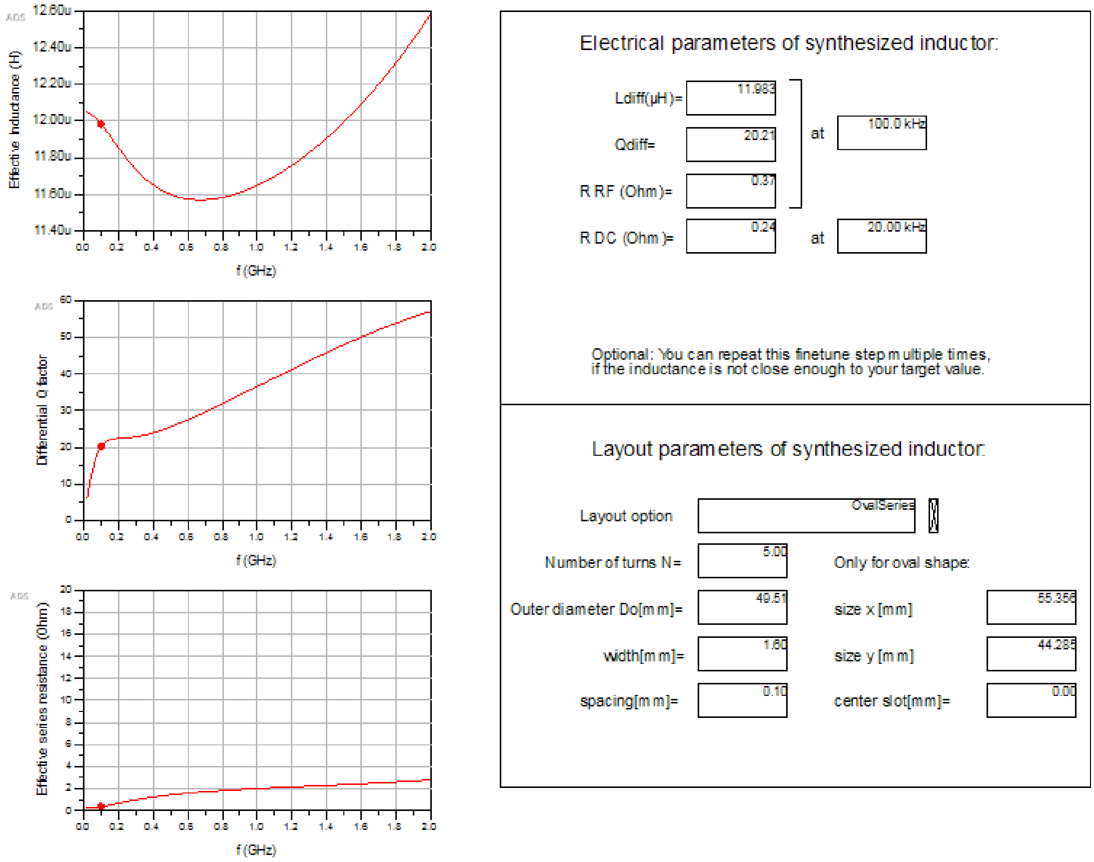


The initial synthesis was based on DC inductance, and the ferrite influence was included with a manual correction factor in the technology file. So that starting value was approximate only, and now we have an actual EM simulated results with the ferrite properly included. At this point there might be some error in inductance, where the EM results from the batch sweep does not meet the target value exactly.

The workflow of Inductor Toolkit is designed to compensate an error in inductance by an additional step, where the inductance is fine tuned by fine tuning the inductor diameter. Based on the best inductors from the batch sweep, this step fine tunes the in ductance by adjusting the inductor diameter. Also, the fine tuning step interpolates the optimum conductor width.

When you click **4: Fine tune inductance value**, this will bring up a simulation schematic that runs one more EM model, with the fine tuned inductor diameter. The dimensions for that tuned layout had been calculated automatically, based on the batch sweep results.

The result of the fine tuning step is the final, optimized inductor. Geometry parameters and electrical parameters are shown in the data display that comes up after simulation:



Comparing to the batch sweep simulation, you can see that the conductor width is unchanged at 1.6mm and the outer diameter was tuned to 49.5mm, to reach the target inductance at the target frequency.

Note on conductor width: The fine tune step can also tweak the conductor width, by interpolation or extrapolation from simulated conductor width values. This is done automatically, if the resulting value is within the geometry limits. In some cases, this might result in inductor diameters that are slightly larger than the specified maximum size.[1]

Iterative fine tune: If the inductance at the taget freqency is not close enough to your specification, you can now repeat the fine tune step, to iterate towards the final layout. In most cases, running the fine tune step once is enough, and there is no need to repeat it.

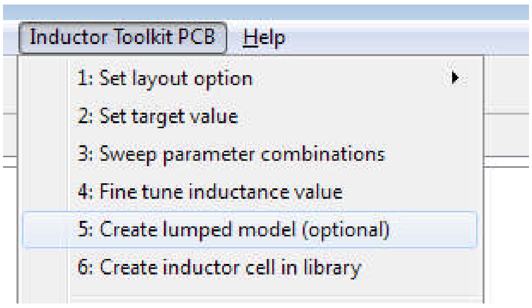
When the fine tune step in finished, we have two possible ways to proceed:

- Create an **equivalent circuit model**, for use in ADS or other simulators, with output as Spectre *.scs netlist file, or
- Directly proceed to the final step, where an **ADS cell** with layout view and schematic view is created.

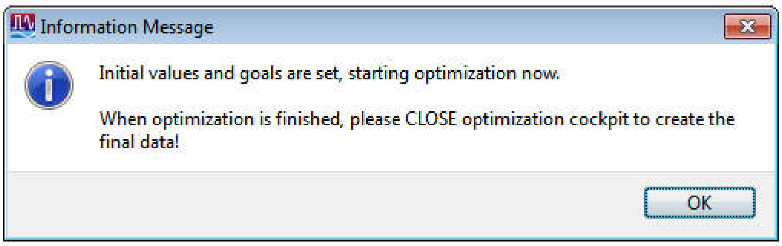
If we skip the equivalent circuit model, the final cell will only have the S-parameters available for schematic level simulation. If we do the equivalent circuit model fit first, the final cell will have two schematic views: one based on the S-parameters, and the other based on the equivalent circuit model.

In this tutorial, we proceed with the optional equivalent circuit model step.

The equivalent circuit model extraction is embedded into the synthesis workflow, and can be done after the fine tune step is finished. The required data is automatically provided by the file tune step.

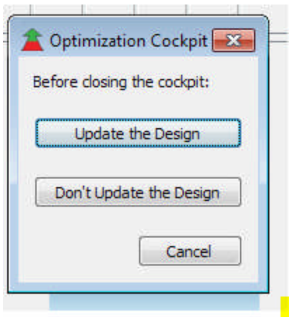


When you click on **5: Create lumped model (optional)**, this will bring up a schematic and automatically start optimization. The schematic is pre-configured with parameters and goals to optimize an equivalent circuit model to your inductor's EM simulation results.



[1] By default, an oversize diameter of up to 105% is tolerated. This can be configured on the equations page of the evaluate_batch_sweep data display.

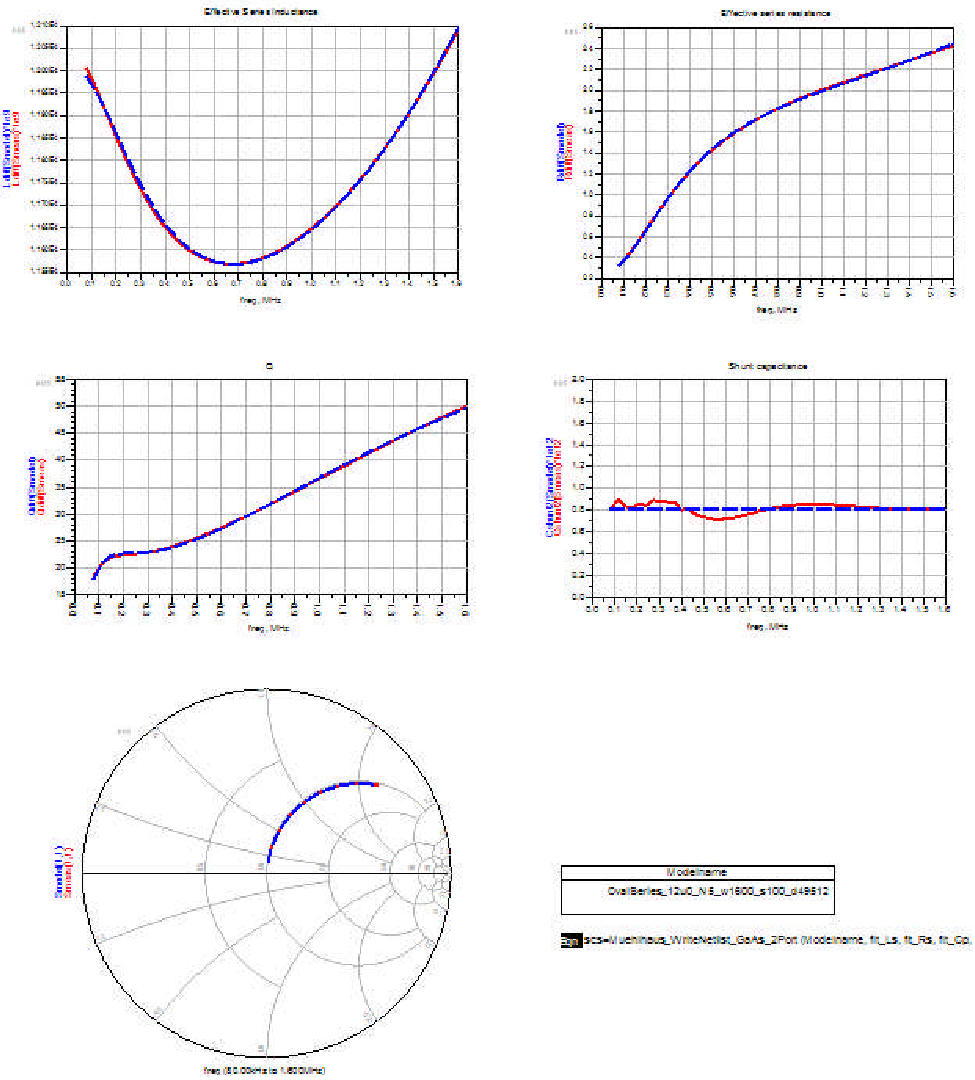
Actually, there are two different equivalent circuit models: with and without center tap. Inductor Toolkit knows what you are working on, and will pick the right one automatically.



The optimization is configured to use the ADS optimization cockpit, so that you can watch the progress of the model fit. When the optimization is finished, **close the optimization cockpit** and confirm that you want to **update the design** with the optimized parameters.

If you stop the optimization manually, before reaching the maximum number of iterations, you have to **manually** start a re-simulation of the optimized schematic. Otherwise, the circuit model fit curves will be empty and no model is created.

red = data from input file
blue with index m = model = fitted data

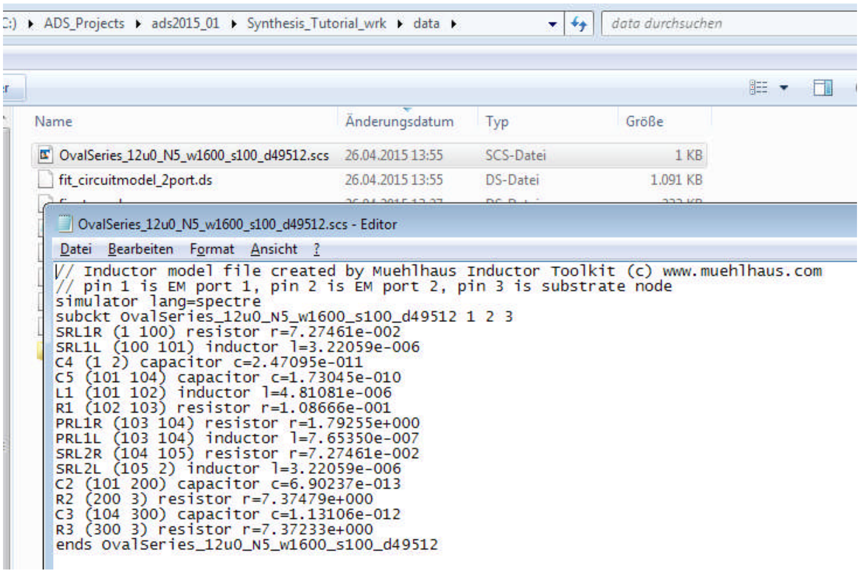


In most cases, you will see excellent agreement between the EM data (red curve) and the fitted circuit model (blue curve) for inductance, Q factor, S-parameters as well as series resistance and shunt path parameters.

If needed, in special cases, you can stop the optimizer and manually tweak the parameter range. You could also tweak the optimizer goals and frequency range, but that's hardly ever needed, because it is all set automatically based on your target frequency and the inductor's electrical parameters.

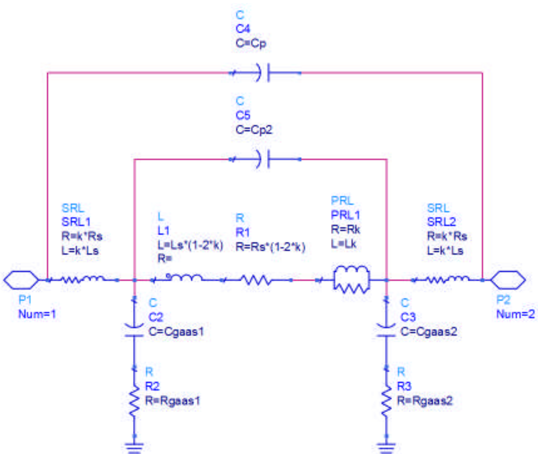
For PCB inductors, the shunt capacitance is usually very small and targer relative error for the shunt path capacitance fit might be seen. This has only very minor effect on the results, and will not cause any problems.

On the bottom right side, you see a list of equivalent circuit model parameters, and a function call that writes the equivalent circuit model to disk, in Spectre *.scs netlist format. That file is created in the workspace's data directory. Looking at the file, you will see the netlist representation of the ADS equivalent circuit model.

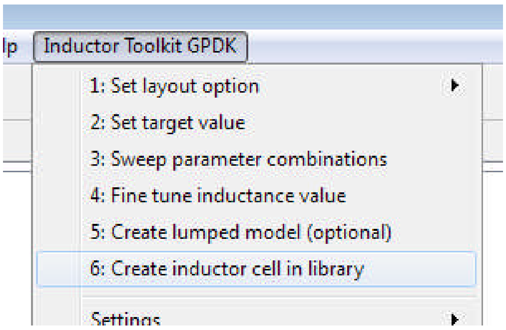



```
// Inductor model file created by Muehlhaus Inductor Toolkit (c) www.muehlhaus.com
// pin 1 is EM port 1, pin 2 is EM port 2, pin 3 is substrate node
simulator lang=spectre
subckt OvalSeries_12u0_N5_w1600_s100_d49512 1 2 3
SRL1R (1 100) resistor r=7.27461e-002
SRL1L (100 101) inductor l=3.22059e-006
C4 (1 2) capacitor c=2.47095e-011
C5 (101 104) capacitor c=1.73045e-010
L1 (101 102) inductor l=4.81081e-006
R1 (102 103) resistor r=1.08666e-001
PRL1R (103 104) resistor r=1.79255e+000
PRL1L (103 104) inductor l=7.65350e-007
SRL2R (104 105) resistor r=7.27461e-002
SRL2L (105 2) inductor l=3.22059e-006
C2 (101 200) capacitor c=6.90237e-013
R2 (200 3) resistor r=7.37479e+000
C3 (104 300) capacitor c=1.13106e-012
R3 (300 3) resistor r=7.37233e+000
ends OvalSeries_12u0_N5_w1600_s100_d49512
```

You can use that netlist in Cadence/Spectre. The topology is fixed, defined by Inductor Toolkit. The circuit model topology can be seen from the circuitmodel_2port.

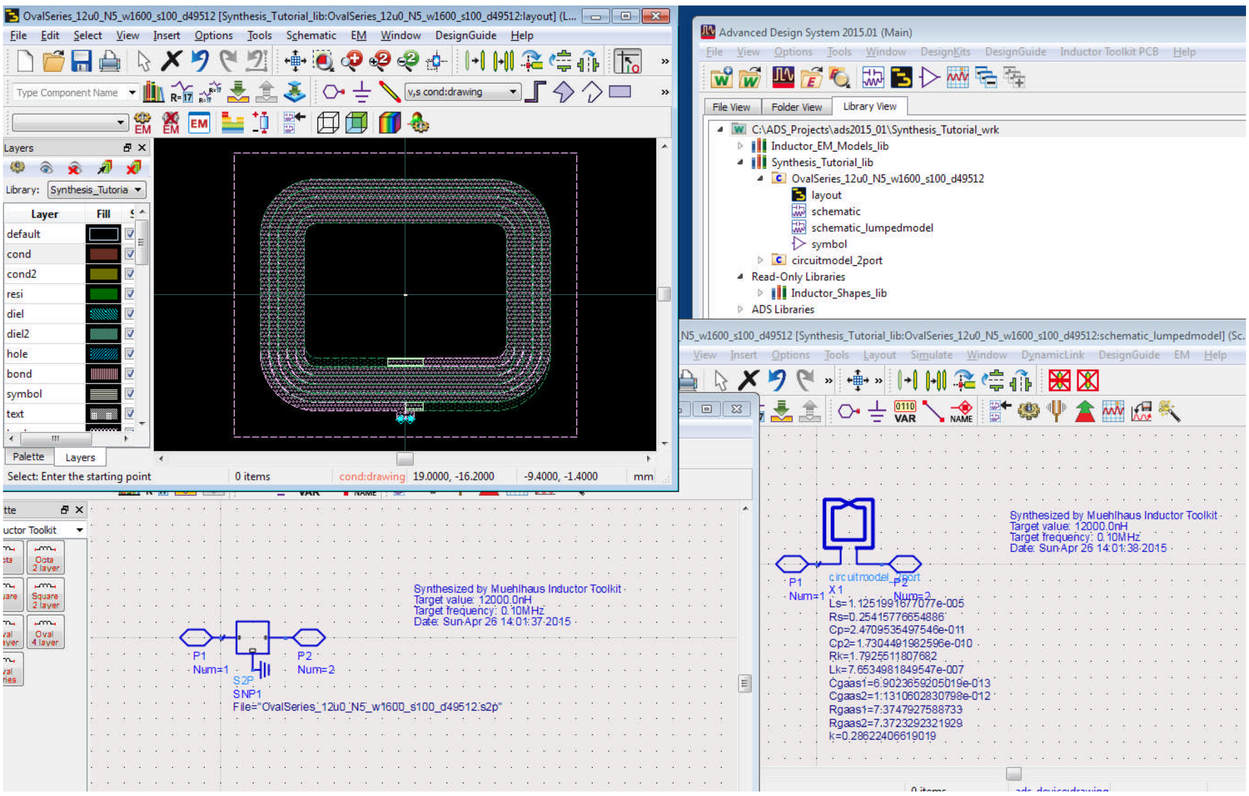


Now, the final step is **6: Create inductor cell in library**.



This will create layout and schematic view(s) of the optimum inductor that we have synthesized, and also create the final S-parameter file in the data directory. The inductor parameters are read from the fine tune synthesis step. The (optional) equivalent circuit model is the optimization result from step 5, Create lumped model.
The cell is created in the first writable workspace library that is not an internal Inductor Toolkit library. In our example, the new cell is created in the Synthesis_Tutorial_lib library.

The name of the cell is derived from the layout option and parameter values. For the PCB example, the optimized inductor cell is Ovalseries_12u0_N5_w1600_s100_d49512 which tells us that this is a differential octagon inductor (without center tap) with 12µH target inductance, N=5 turns, w=1600µm conductor width, s=100µm conductor spacing and 49512µm outer diameter.



For the actual x and y dimensions, that diameter must be multiplied with SQRT(aspect ratio) for the x dimension, and divided by SQRT(aspect ratio) for the y dimension.

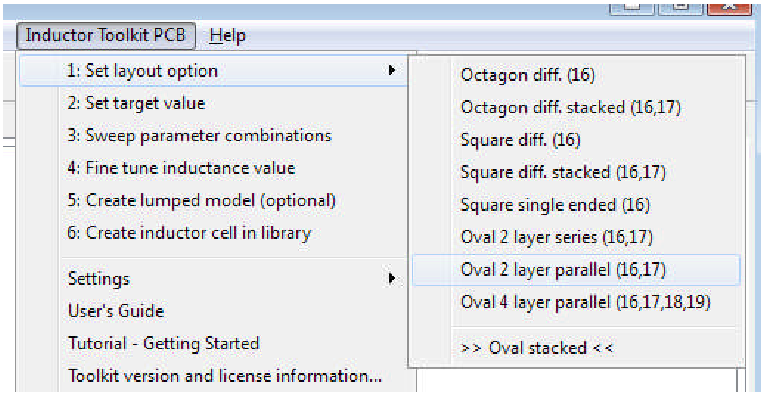
In the layout view, we have an instance of the differential octagon inductor pcell, with these geometry parameters. By default, it is not flattened to polygons.

S-parameter and *.scs Spectre model file have been created in the workspace's data directory, with base name Ovalseries_12u0_N5_w1600_s100_d49512.

Second example: stacked metal and slot

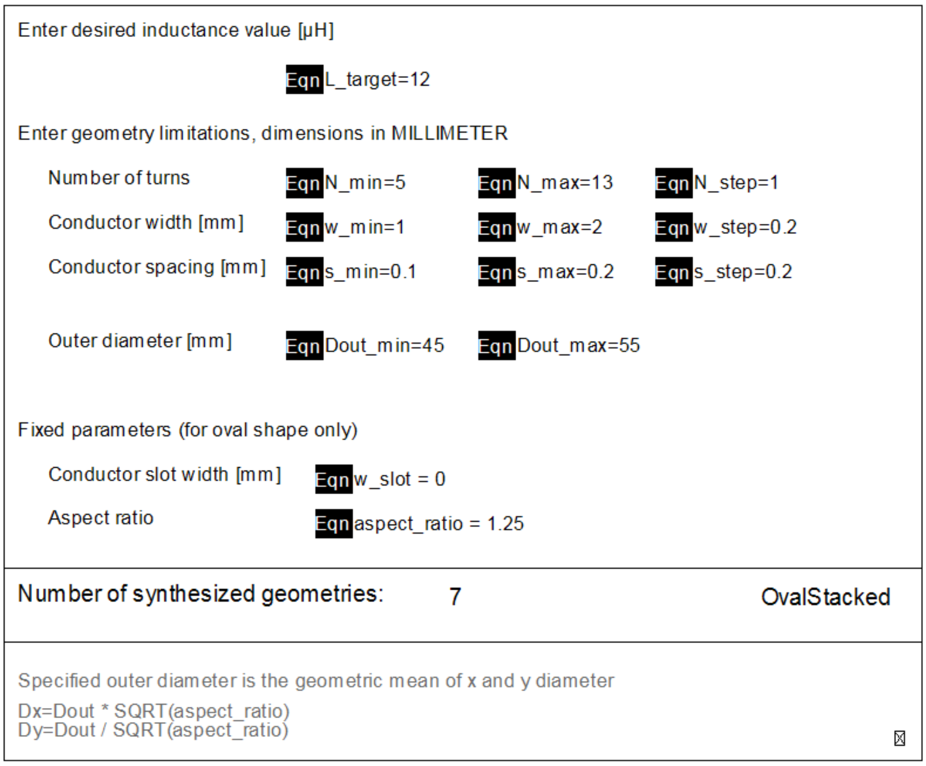
In this chapter, we want to investigate the stacked metal oval inductor, with two stacked metals. This requires more turns for the same inductance, because the two layers are not connected in series.

First, select the inductor shape in the Inductor Toolkit menu, **step 1**.



Then, select **step 2: Set target value.** This opens the data display form again where you can modify the geometry limitations.

Change N_max to 13 and change s_max to 0.2. Notice that the number of synthesized geometries is now 7.

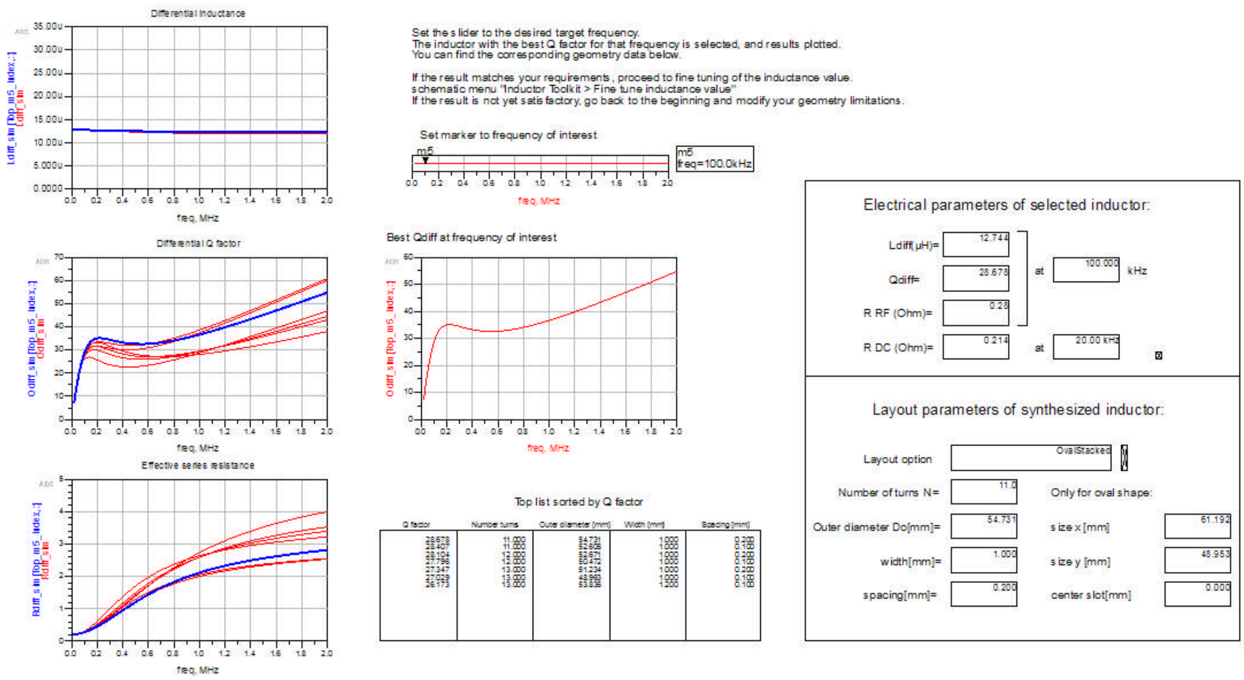


Save your changes, so that we can check the geometry choices later.

Let's see how this inductor type performs. In the Inductor Toolkit menu, click on **3: Sweep parameter combinations**. This will start the batch simulation of all the 7 different layouts.

Simulation starts and again, it takes a while to EM simulate the different parameter combinations.

So what about results? If you had saved the data display in tutorial example 1, it will remember the target frequency. Otherwise, please set it to 100kHz again.



Now, we reach a Q factor of 28.7 at 100kHz, with 11 turns and 1mm trace width at 0.2mm spacing. This is better than the Q=20 we had found for the oval series layout.

The different results can be compared in the inductor top list at the bottom of the results page. We see that 11 turns performs best, and 0.2mm spacing is slightly better than 0.1mm spacing.

Top list sorted by Q factor

Q factor	Number turns	Outer diameter [mm]	Width [mm]	Spacing [mm]
28.678	11.000	54.731	1.000	0.200
28.407	11.000	52.606	1.000	0.100
28.104	12.000	52.671	1.000	0.200
27.796	12.000	50.472	1.000	0.100
27.347	13.000	51.234	1.000	0.200
27.029	13.000	48.963	1.000	0.100
26.173	13.000	53.836	1.200	0.100

We can see that our trace width is at the lower limit of the range that we specified. This is because we need a certain number of turns to reach the target inductance, and wider metal traces would result in too large inductors (outside specified diameter range).

If we want to keep the outer dimension limits, and check for possibly better inductors, we can run the batch again with the lower limit changed to 0.6mm, so that we check for possibly better layouts in that extended range.

If we decide to do that, we find that the best width is still 1mm, and the 0.8mm width inductors all have lower Q factor. The previous best inductor parameters are still the best, after extending the geometry range to smaller widths.

Top list sorted by Q factor

Q factor	Number turns	Outer diameter [mm]	Width [mm]	Spacing [mm]
28.678	11.000	54.731	1.000	0.200
28.407	11.000	52.606	1.000	0.100
28.104	12.000	52.671	1.000	0.200
27.796	12.000	50.472	1.000	0.100
27.347	13.000	51.234	1.000	0.200
27.029	13.000	48.963	1.000	0.100
26.857	10.000	50.911	0.800	0.100
26.751	10.000	53.061	0.800	0.200
26.734	11.000	49.999	0.800	0.200
26.729	11.000	47.772	0.800	0.100
26.571	12.000	45.518	0.800	0.100
26.556	12.000	47.818	0.800	0.200
26.277	13.000	46.258	0.800	0.200

If we don't increase the diameter, there is no better parameter combination for this layout. The only possible alternative would be to now include a center slot in the inductor. The slot width is set to 0.2mm (was 0.0 before) and the minimum width is changed back to 1mm.

Enter desired inductance value [µH]

Eqn L_target=12

Enter geometry limitations, dimensions in MILLIMETER

Number of turns	Eqn N_min=5	Eqn N_max=13	Eqn N_step=1
Conductor width [mm]	Eqn w_min=1	Eqn w_max=2	Eqn w_step=0.2
Conductor spacing [mm]	Eqn s_min=0.1	Eqn s_max=0.2	Eqn s_step=0.2
Outer diameter [mm]	Eqn Dout_min=45	Eqn Dout_max=55	

Fixed parameters (for oval shape only)

Conductor slot width [mm] Eqn w_slot = 0.2

Aspect ratio Eqn aspect_ratio = 1.25

Number of synthesized geometries: 7 OvalStacked

Specified outer diameter is the geometric mean of x and y diameter

Dx=Dout * SQRT(aspect_ratio)
Dy=Dout / SQRT(aspect_ratio)

After making that change, batch simulation (step 3) is started again, and now runs the inductors with slot.

Results for the simulation with 0.2mm slot show that the Q factor has **decreased** from 28 to less than 24. There is no benefit from using the slot for this layout, and it performs **worse** than the solid conductor version. Adding a slot means decreasing the metal cross section, which is not always desirable.

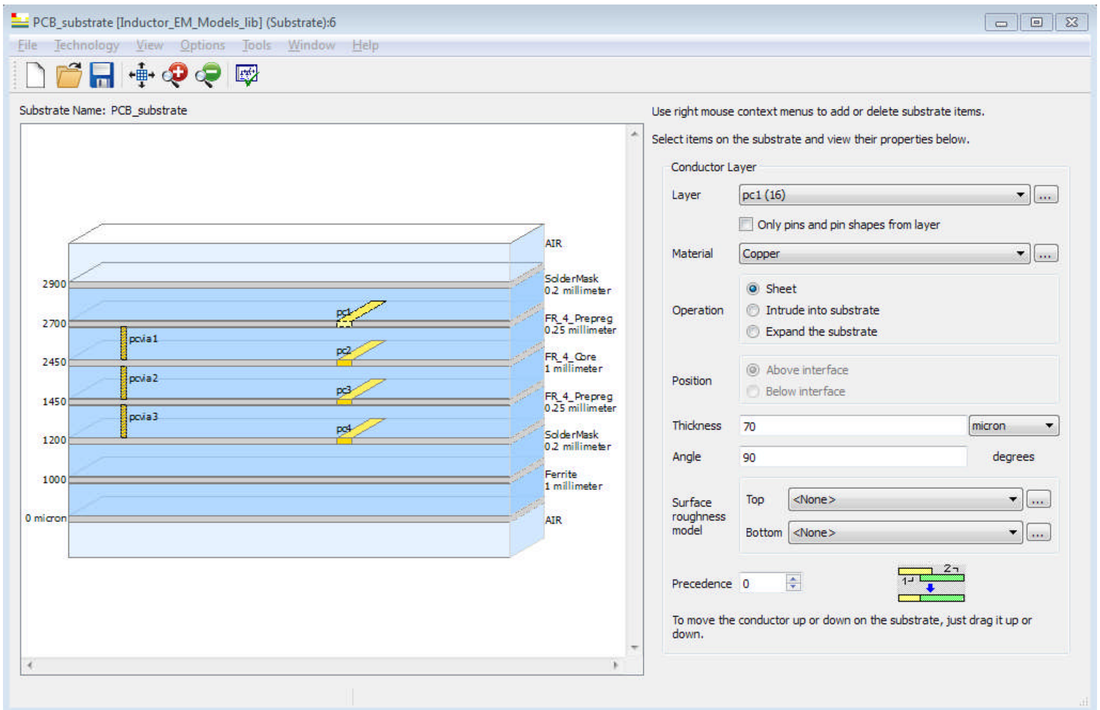
Top list sorted by Q factor

Q factor	Number turns	Outer diameter [mm]	Width [mm]	Spacing [mm]
23.851	11.000	54.731	1.000	0.200
23.679	11.000	52.606	1.000	0.100
23.428	12.000	52.671	1.000	0.200
23.202	12.000	50.472	1.000	0.100
22.886	13.000	53.836	1.200	0.100
22.830	13.000	51.234	1.000	0.200
22.581	13.000	48.963	1.000	0.100

Technology: Editing the EM substrate file

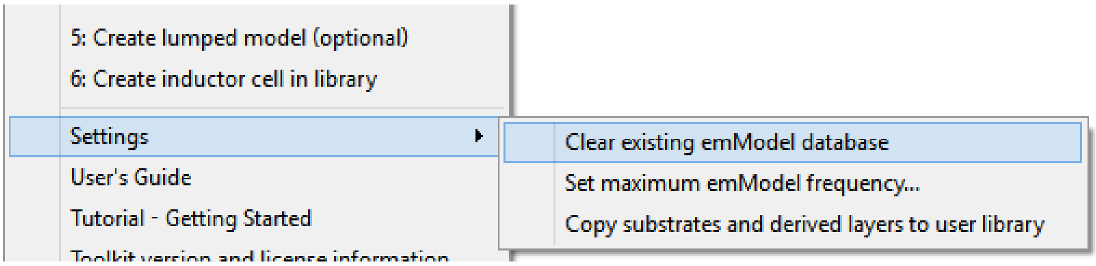
For simulation, Inductor Toolkt uses the substrate "PCB_substrate" from library "Inductor_EM_Models_lib".

The PCB stackup, including the ferrite, can be changed according to your actual technology in the Momentum substrate editor. The pre-defined Momentum substrate file uses 70µm metal thickness, simulated as flat sheets. The core dielectric is defined with 1mm thickness between the metals, the outer dieletric thickness with 0.25mm between the metals and the solder mask is defined as 0.2mm.



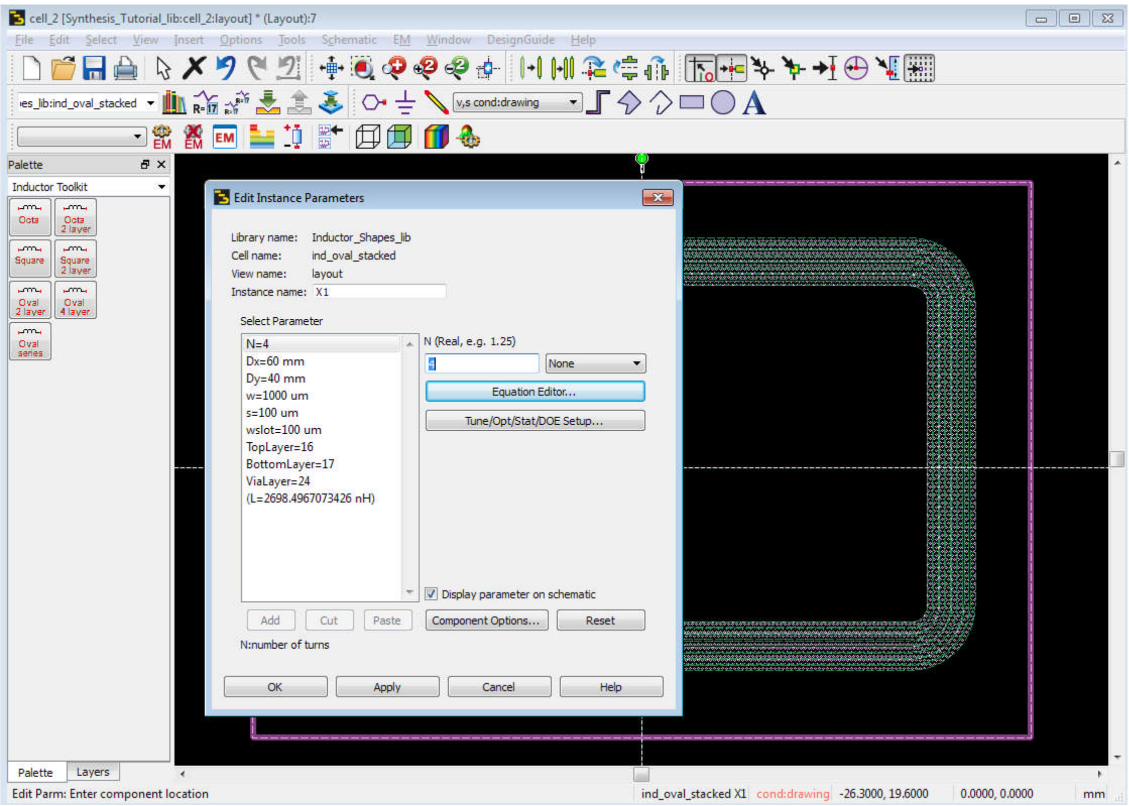
The thin metal model includes the conductor losses, but does not include the change of field distribution from metal thickness. Please refer to the User's Guide, chapter "Changing the substrate definition" for more information on thick metal simulation.

IMPORTANT: After making ANY changes to the substrate defintion, you MUST ALWAYS clear the emModel cache data that has become invalid. This is done in the Inductor Toolkit menu: Settings > Clear existing emModel database.

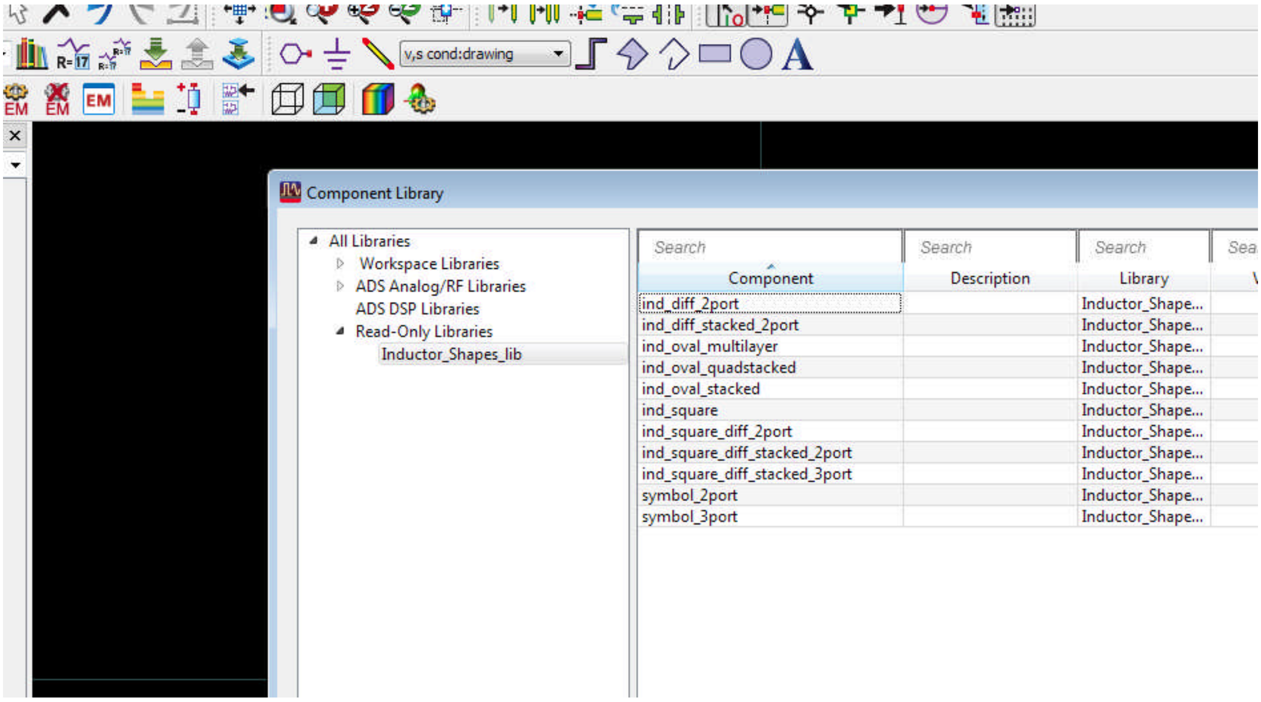


Using the layout Pcells for manual layout

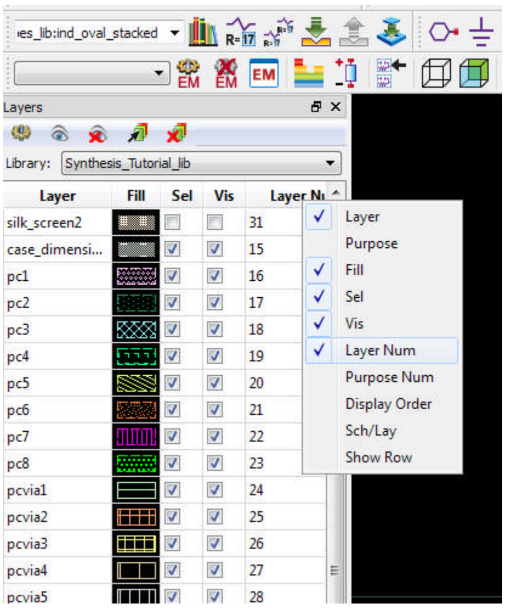
You can use the parameterized layout cells for manual layout also. Place a layout cell by selecting it from the Inductor Toolkit component palette in layout.



An alternative is to use drag & drop from the Inductor_Shapes library in the project tree view, or to select the layout from the component library. The cell names and layouts are described in the Inductor Toolkit User's Guide.



Note that the oval shapes are specified by x and y outer dimensions when the cell is placed in layout manually (synthesis uses geometric mean diameter and aspect ratio).
You can set the layers as needed. Layers are defined by the ADS layer number. For convenience, you can enable the layer number display in the ADS layers window: right click on the title bar and enable that column.



This ends the tutorial.

Changes

Document version 1.0 of 27. April 2015: Initial version