Application Report

Simulate Inductive Sensors Using FEMM (Finite Element Method Magnetics)

ABSTRACT

When determining the resolution of an inductive sensing system, it can be especially helpful to simulate the response of the system before building the sensor. Finite Element Method Magnetics (FEMM) is open-source software that can simulate electromagnetic and electrostatic problems. This application note is a step-by-step guide for using FEMM to simulate inductive sensing solutions using LDC devices and estimate system resolution using the simulation results. This application note is applicable to the LDC3114, LDC161x, LDC131x, and LDC1101 devices.

Table of Contents

List of Figures

List of Tables

Trademarks

All trademarks are the property of their respective owners.

Copyright © 2021 Texas Instruments Incorporated

1 Introduction

Simulation tools similar to SPICE are an important part of the design process for many systems because they reveal design problems before prototypes are built. SPICE, however, is not suitable to simulate inductive sensing systems because it does not include target interactions with the sensor coil. The open-source magnetics simulation software FEMM (Finite Element Method Magnetics) can provide information about these target interactions and the robustness of the system before production. Before using FEMM, we recommend using the LDC Tools Excel calculator to design the sensor coil, as well as reading LDC Sensor Design and LDC Target Design. For smooth targets that are at least as large as the sensor coil, TI also recommends using the Excel_FEMM tool (included in LDC Tools), which will automatically simulate custom sensors in FEMM. For smaller or irregularly shaped targets, the following guide will give the reader working knowledge of FEMM for inductive sensing simulations. The program can be downloaded from the FEMM home page.

2 Inductive Sensor Simulation

1. Design a coil using the LDC Tools Excel calculator's *Spiral Inductor Designer*. The dimensions of this coil will be used to create the FEMM simulation. Instructions are embedded in the tool.
2. Create a new problem
   a. Open FEMM
   b. Click **File** → **New**
   c. In the **Create a new problem** dialog box, select **Magnetics Problem** and click **OK**.

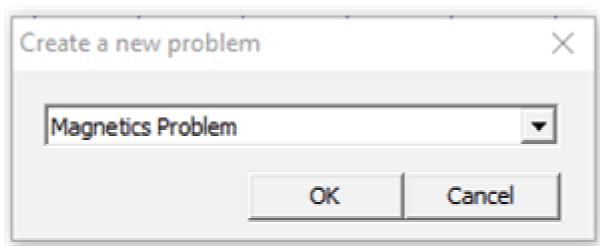


Figure 2-1. Create a New Problem Dialog Box

3. Make the new problem a planar problem or an axisymmetric problem.

   When choosing between the two problem types, consider the geometry of the coil. For both planar and axisymmetric problems, the FEMM workspace forms a cross section of the coil and target. The workspace is a slice of the PCB coil such that all layers and traces of the PCB are visible. Keep this in mind while constructing the coil. It is important to construct the cross section such that the simulation will create a coil shape. See how each problem type turns the 2D workspace into a 3D solution:

Copyright © 2021 Texas Instruments Incorporated

a. Planar problems:

Planar problems expand along the axis perpendicular to the plane. The user must specify the distance of this expansion. This type of problem is most often used to simulate racetrack coils. However, planar problems cut off the rounded edges of the coils, leaving only the long parallel section. This makes the simulation much less accurate than spiral coil simulations using axisymmetric problems.

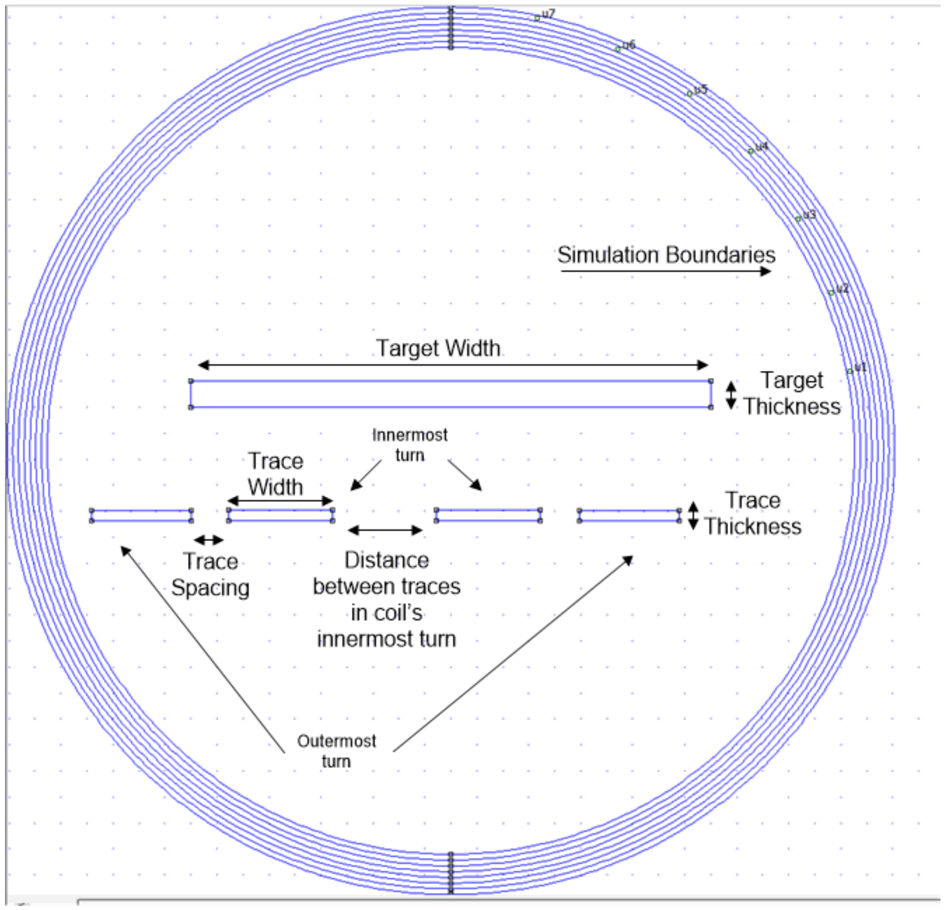


Figure 2-2. Example Planar Problem Workspace With Labeled Traces and Target

Copyright © 2021 Texas Instruments Incorporated

b. Axisymmetric problems:

Axisymmetric problems rotate around a user-specified axis. This type of problem is the most useful for inductive sensing simulations and should be used for all circular coils. Note that coils simulated with this problem type will not be true coils, but will be a series of concentric circles. Even so, the coils are fairly accurate simulations.

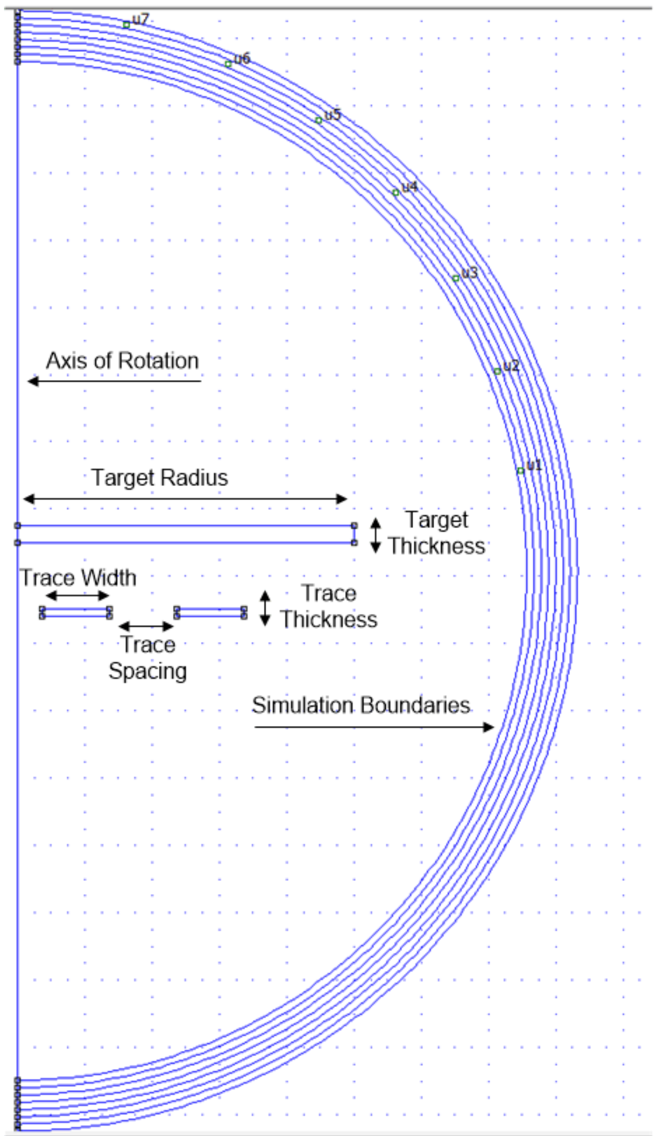


Figure 2-3. Example Axisymmetric Problem Workspace With Labeled Traces and Target

4. Define the problem.
   a. Navigate to the **Problem** menu shown in Figure 2-4.
   b. In the **Problem Definition** dialog box, choose planar or axisymmetric for the **Problem Type**. If “planar” is selected, enter the depth of the problem expansion in the **Depth** field.
   c. Enter the appropriate length units for the coil design in the **Length Units** field. Typically mm or mils are most useful.
   d. Define a Frequency (Hz). All magnetic problems must have a defined frequency. For inductive sensing problems, this is comparable to choosing a capacitor for a known inductor.

      Use the **Spiral Inductor Designer** in **LDC Tools** to calculate a frequency and enter that frequency in the problem **Frequency (Hz)** field. If the FEMM simulation includes a target, use the sensor frequency with target interaction.
   e. In the **AC Solver** field, select **Succ. Approx**. Ignore the other fields and click **OK**.

Copyright © 2021 Texas Instruments Incorporated

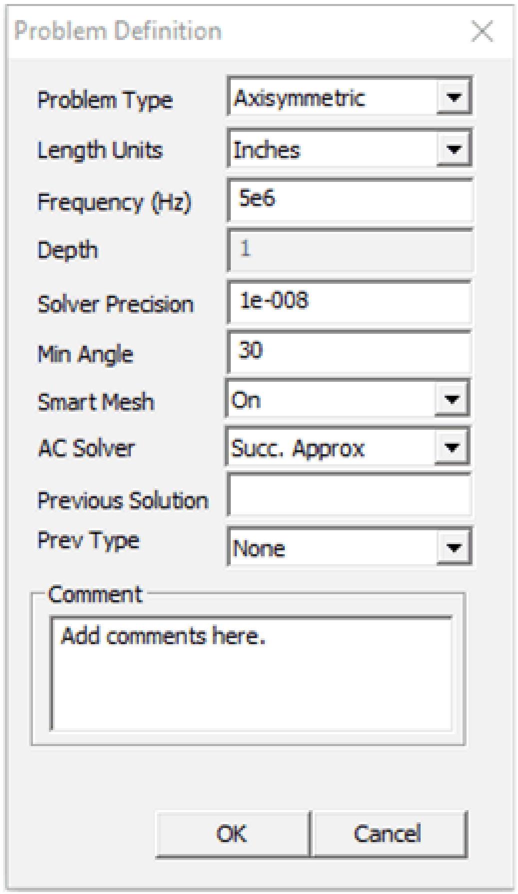
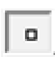


Figure 2-4. Problem Definition Dialogue Box

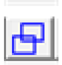
5. Place nodes

   a. Use this button  to place nodes. Use either segments  or arcs  to connect them. At least two nodes must be placed before placing segments. These will form the boundaries of your shape(s).

      Use the trace width, trace spacing, and trace thickness from the **Spiral Inductor Designer** to create a cross section of each trace. Here are key tips for placing objects:

      i. It can be challenging to place nodes exactly in-line with each other. It may be easier to place one node, copy it, and specify X- and Y- translations for the new nodes. Note that the units are specified in the **Problem Definition** menu.
      ii. Click close to the desired nodes to place segments or arcs on the node.
      iii. Though the coils in this tutorial will use rectangular traces, the best approximation of real traces is a trapezoidal shape.

   b. Other helpful tips when moving objects:

      i. Right-click near an object to select it.
      ii. Double-right-click the object to show the X,Y position of the object.
      iii. Navigate with keyboard arrows.
      iv. Click the Page Up and Page Down keys to zoom in and out.
      v. Undo only works ONCE.
      vi. Click the  button to move objects.
      vii. Click the  button to copy objects.

Copyright © 2021 Texas Instruments Incorporated

- viii. Click the button to select a group of objects. Then click the button to highlight objects in a rectangular area or the button to highlight objects in a circular area. Press the space bar to join the objects into a group.
- ix. Click the button to scale

c. For axisymmetric problems only: Mouse over the leftmost segment or arc or double-right-click the arc and look at the bottom left corner of the screen to determine the X,Y location of the object. Select all drawn objects and translate them left so that the leftmost segment or arc now has an x-value equal to the inner radius of the sensor coil. The axis of rotation will be the y-axis, so offsets in the x-direction will form gaps or holes in the object.

6. Define boundaries

a. Click the button to open the **Boundary Builder**.

b. The **Boundary Builder** builds the model, shows the axis of symmetry, and defines several boundaries that form a gradient to infinity.

c. Leave the values in the **Open Boundary Builder** dialog box unchanged and click **OK**.

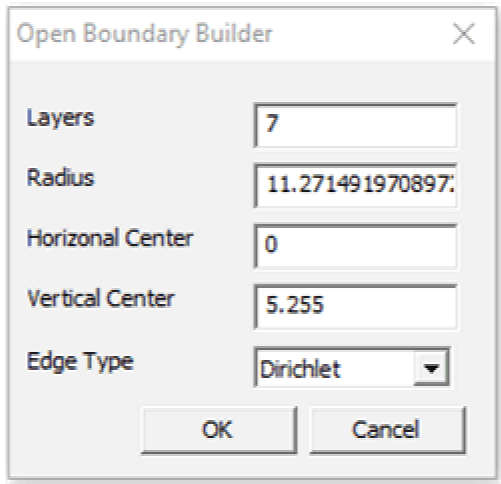


Figure 2-5. Boundary Builder Dialogue Box

d. The workspace should now look similar to Figure 2-6 for axisymmetric problems or similar to Figure 2-7 for planar problems:

Copyright © 2021 Texas Instruments Incorporated

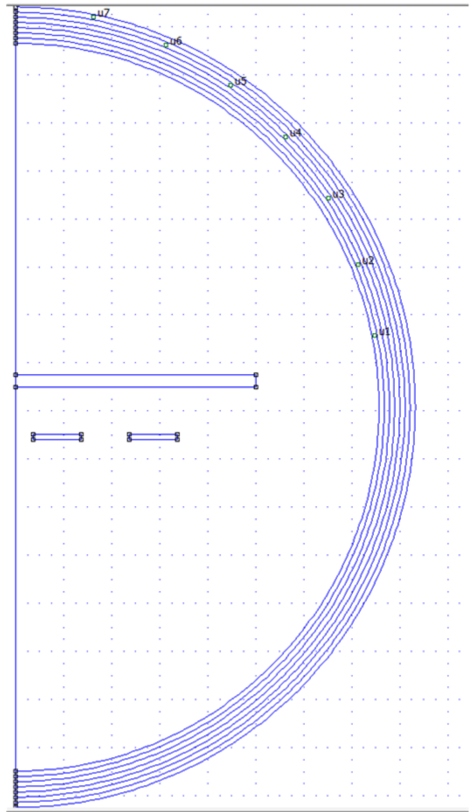


Figure 2-6. Axisymmetric Problem With Boundaries

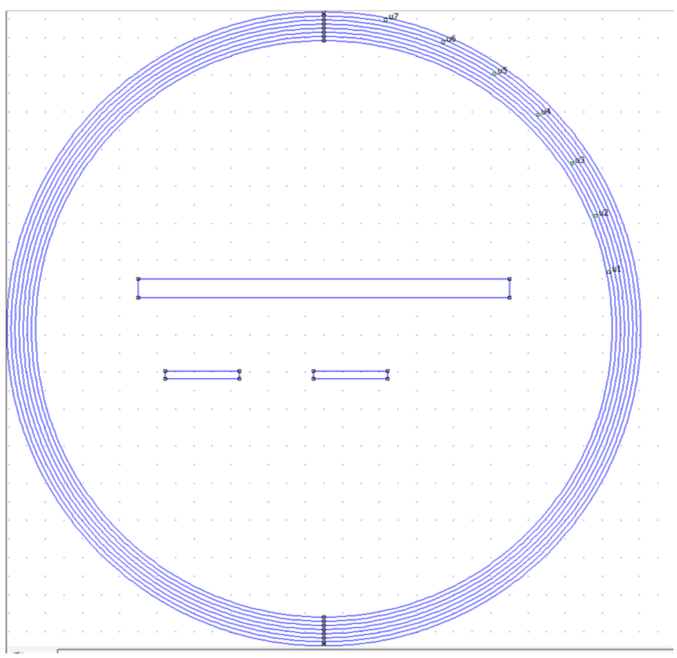
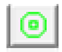


Figure 2-7. Planar Problem With Boundaries

Copyright © 2021 Texas Instruments Incorporated

7. Define materials

   a. Select the circular green button  to place block labels. Place a block label inside each boundary area, including the white space inside the outer boundary. The result should look something like Figure 2-8:

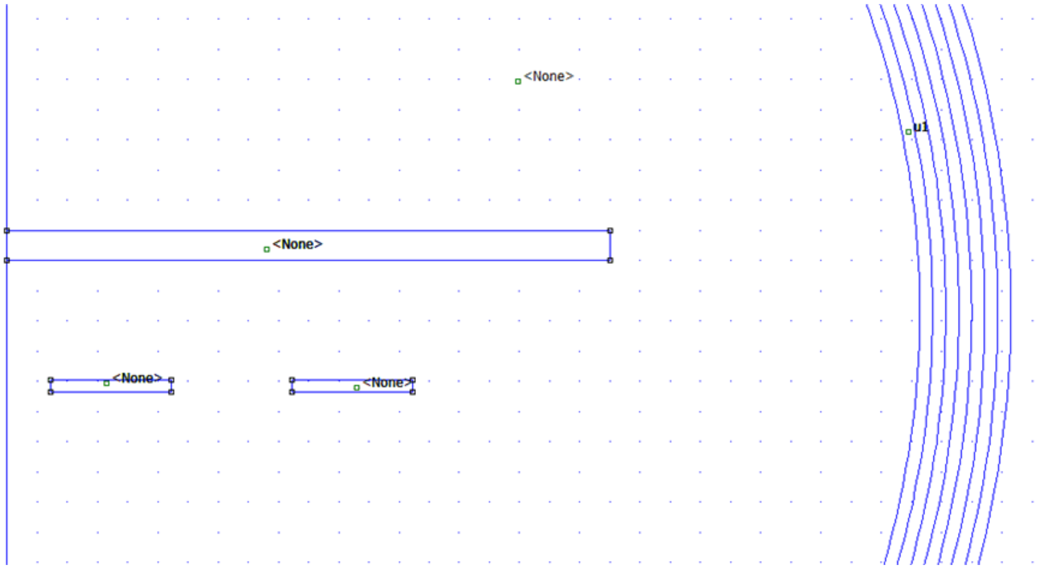


Figure 2-8. Axisymmetric Problem Workspace With Block Labels

   b. Each block label must be changed from <None> to a defined material. First, the materials must be added to the Materials Library. Navigate to **Properties → Materials Library.**

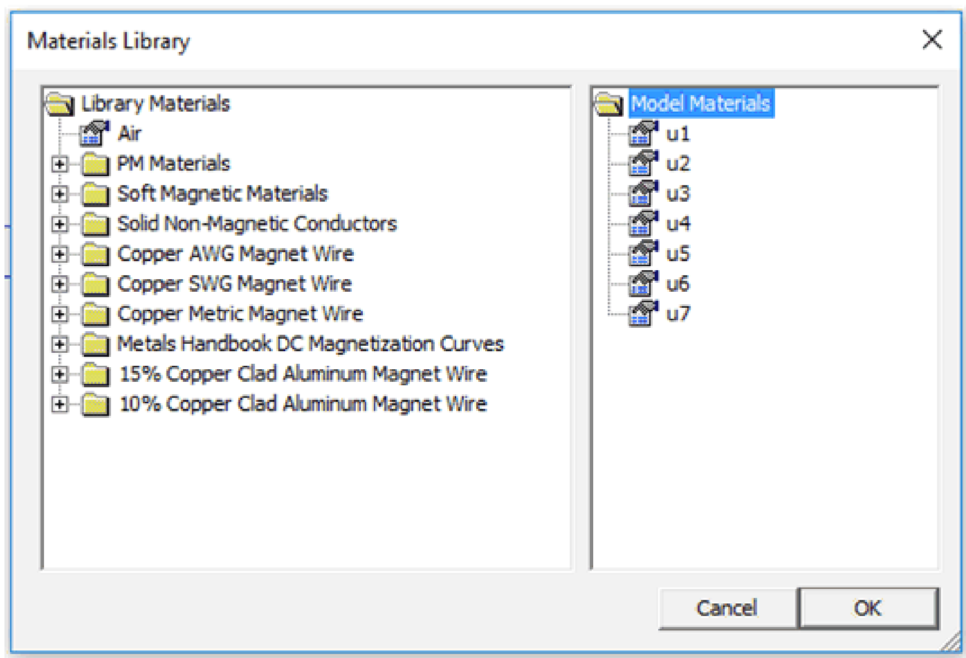


Figure 2-9. Default Materials Library

   c. Typically, the engineer only uses air, copper, and the target material if the material is not copper. Drag **Air** from the left side to the right side of the menu. Open the **Solid Non-Magnetic Conductors** folder and drag **Copper** to the right side of the menu. Click **OK**.

Copyright © 2021 Texas Instruments Incorporated

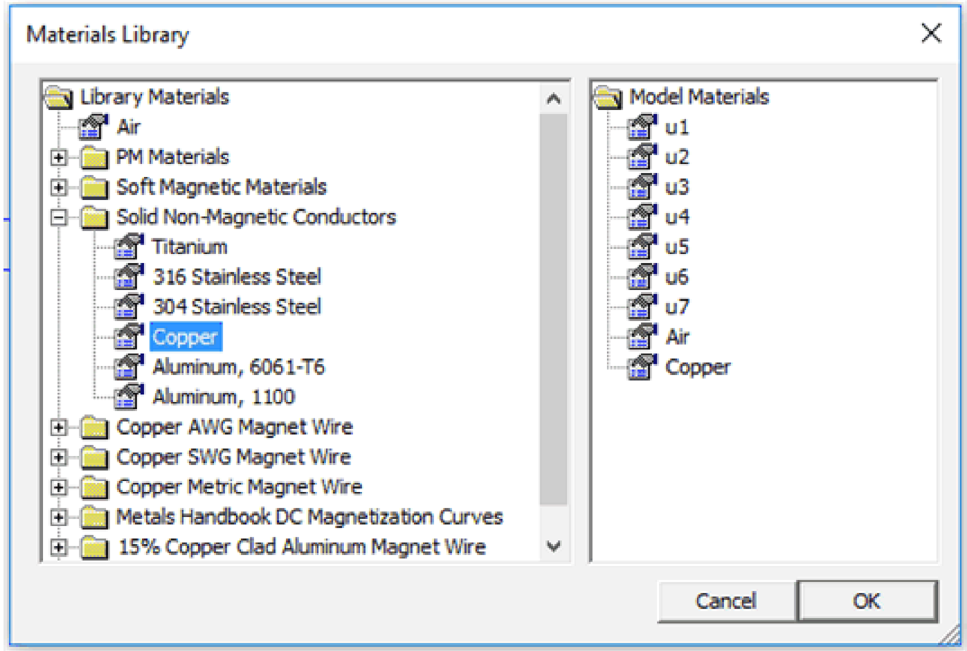


Figure 2-10. Materials Library With Air and Copper Added to Model Materials

d. The copper coil requires a special material type. Navigate to **Properties** → **Materials**. In the **Property Definition** menu, click **Add Property** to open the **Block Property** dialog box.

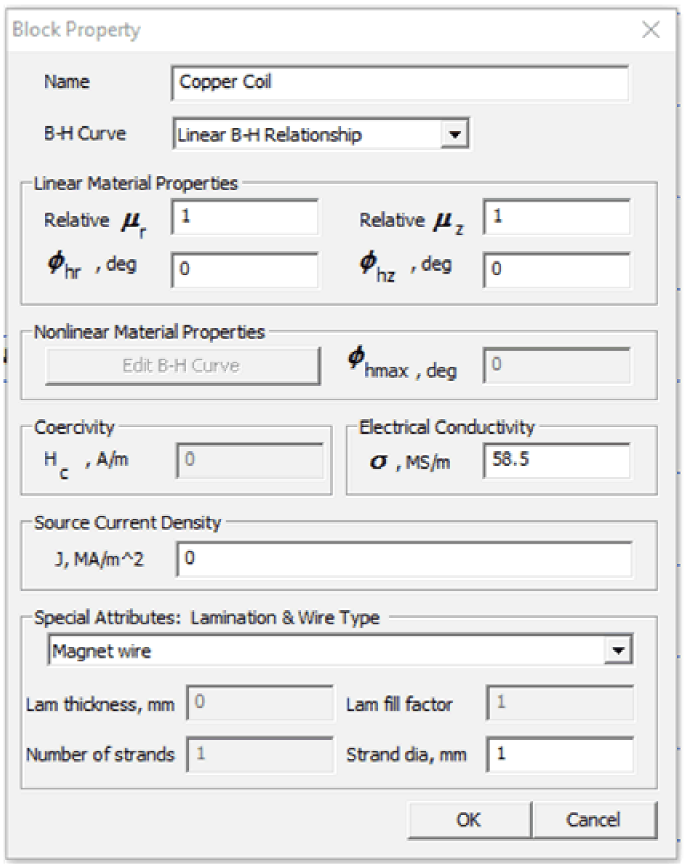


Figure 2-11. Block Properties Menu

e. i. In the **Name** field, enter the material name **Copper Coil**.
ii. In the **B-H Curve** drop-down menu, select **Linear B-H Relationship**.
iii. The **Linear Material Properties** do not need to be modified. Copper is a conductor, therefore the material will have a relative permeability of 1.

Copyright © 2021 Texas Instruments Incorporated

iv. Change the **Electrical Conductivity** field to **58.5 MS/m**.
v. Change the **Special Attributes: Lamination & Wire Type** to **Magnet wire**.
vi. In the **Strand dia, mm** field, enter the strand diameter based on the trace width and copper thickness of the real coil. Use Equation 1 to calculate the equivalent strand diameter:
   1. The trace height and width form the cross section of the trace. The area of the equivalent copper wire must equal the cross sectional area of the PCB trace.
   2. Solve for r in mm. Use 2r for the strand diameter and enter this value in the **Strand dia, mm** field.
   3. This calculated diameter does not affect the size of the inductor nor the spacing. This is determined by the boundaries in the workspace. The strand diameter does affect the coil's series resistance.

$$TW = \pi r^2 \tag{1}$$

- T is the trace thickness in mm
- W is the trace width in mm
- r is the strand radius in mm

f. Right-click near the material labels to highlight the names. Press the space bar to open the **Properties for selected block** menu. Change the Block type to the appropriate material for the area. Repeat this for all material labels.

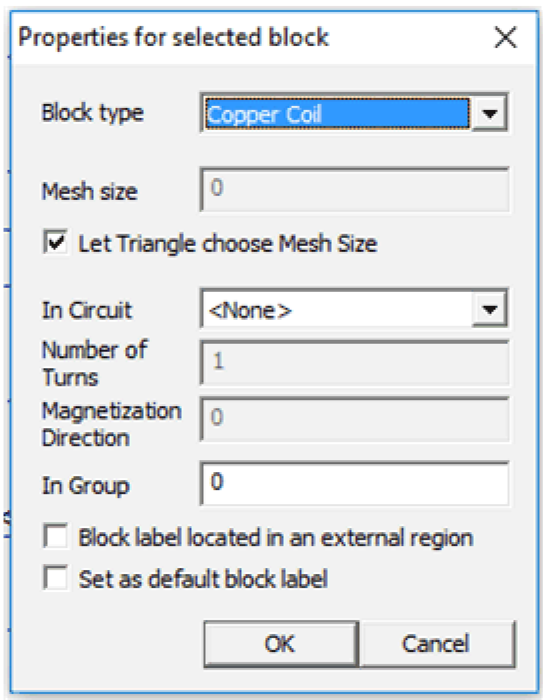


Figure 2-12. Properties for Selected Block Menu

g. After these steps, the workspace should look something like Figure 2-13:

Copyright © 2021 Texas Instruments Incorporated

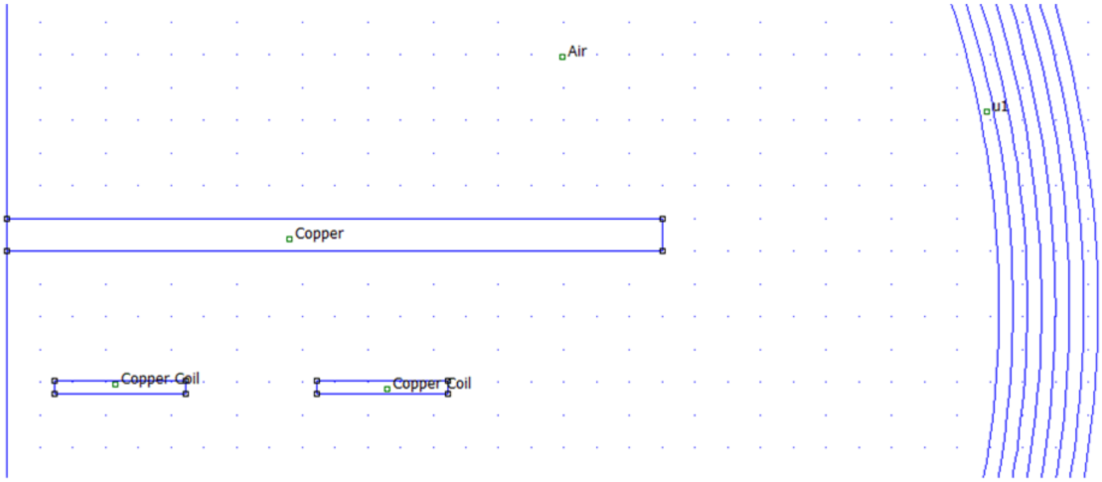


Figure 2-13. Axisymmetric Problem Workspace With Defined Materials

8. Add circuits to the problem
   a. Navigate to the **Properties** → **Circuits** menu. In the **Property Definition** dialog dox, click **Add Property**.
   b. In the **Circuit Property** dialog box, name the circuit and specify an operating current.
      i. The current value is not especially important. A value anywhere from 1 mA to 1 A is acceptable.
   c. Make sure the **Series** option is checked and click **OK**.

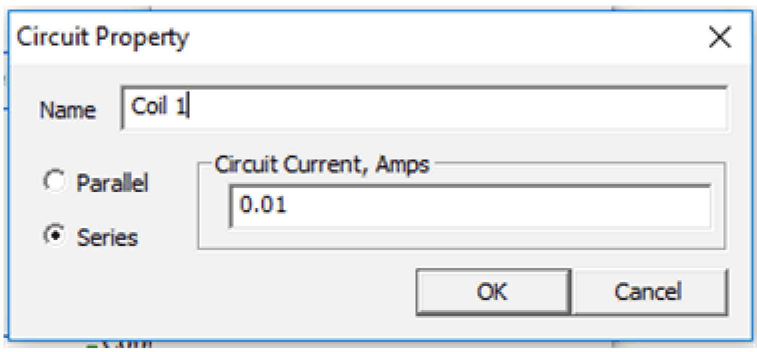


Figure 2-14. Circuit Property Menu

   d. For planar problems only:
      i. For racetrack coils and similar circuits, the parallel traces are not automatically connected by the planar expansion. At least two separate circuits are required to correctly model the current flow.
      ii. To model the current flow, define two circuits with the same current magnitude. One should be positive and the other should be negative. Assign one to each side of the racetrack coil. Your workspace should look similar to Figure 2-15:

Copyright © 2021 Texas Instruments Incorporated

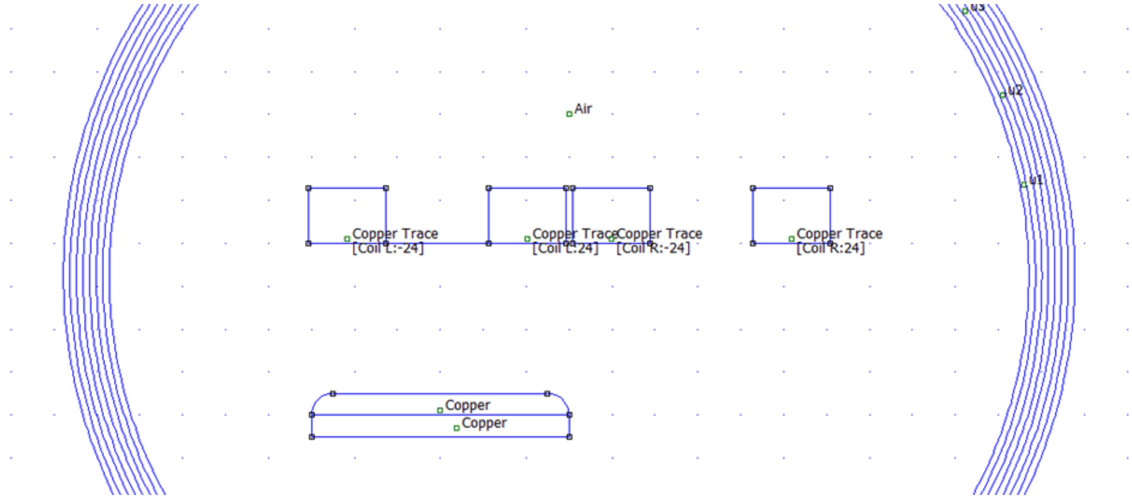


Figure 2-15. Example Planar Problem Workspace With Defined Circuits for Racetrack Coil

e. Right-click the material nodes again and click the space bar to return to the **Properties for selected block** menu.

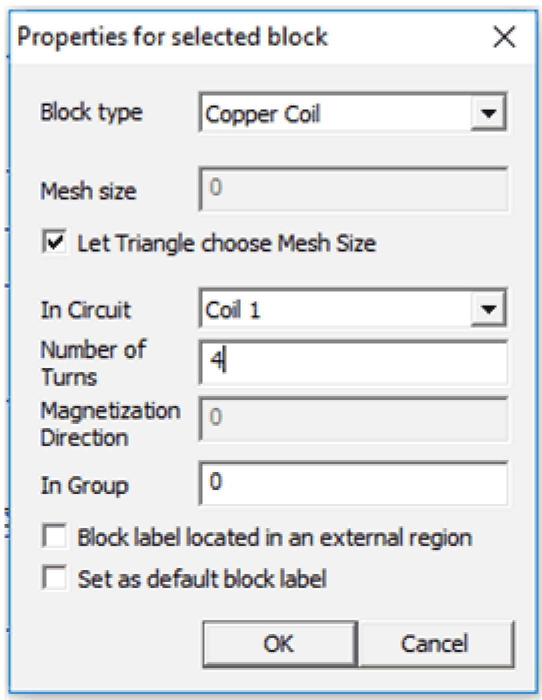


Figure 2-16. Properties for Selected Block Menu

f. i. Select the circuit that you just created in the **In Circuit** field.
   ii. Specify the number of turns in the coil. This will not change the size of the coil and only change the generated magnetic field. The simulation will yield the most realistic results if only one turn is used per coil boundary and the number of coil boundaries is equal to the number of turns in the real coil.
   iii. Leave the **Let Triangle choose Mesh Size** box selected.

g. Repeat this process for all boundaries in the coil. The workspace should now look similar to Figure 2-17:

Copyright © 2021 Texas Instruments Incorporated

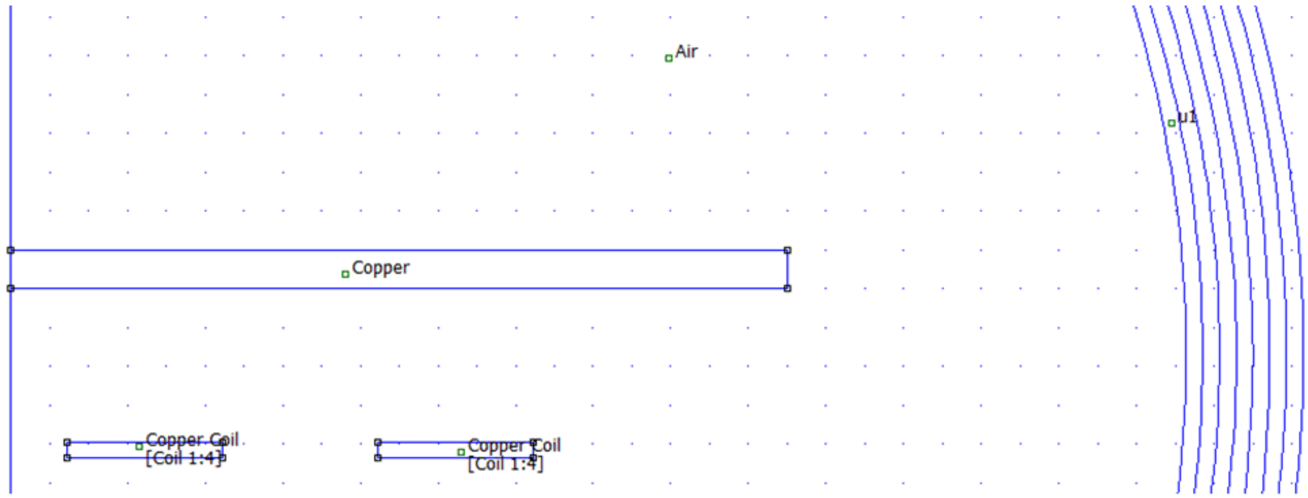


Figure 2-17. Axisymmetric Problem Workspace With Defined Circuits

9. Generate the answer file
    a. Click the **Solve** button  to generate the answer file.
    b. Answer files will be saved in the .ans file format in the current working directory.

Copyright © 2021 Texas Instruments Incorporated

3 Using the Electromagnetic Problem Answer File

1. Open the .ans file
   a. The .ans file will generate in the same directory as the .FEM file. Open the .ans file.
   b. Note that if any changes are made to the .FEM file, solving the problem again will overwrite the .ans file.
   c. The .ans file should show the magnetic field lines, which should bend around the target. The outer boundaries represent a gradient to infinity, so the magnetic field lines may contort in this region. Only the region contained by the innermost boundary line is of interest.

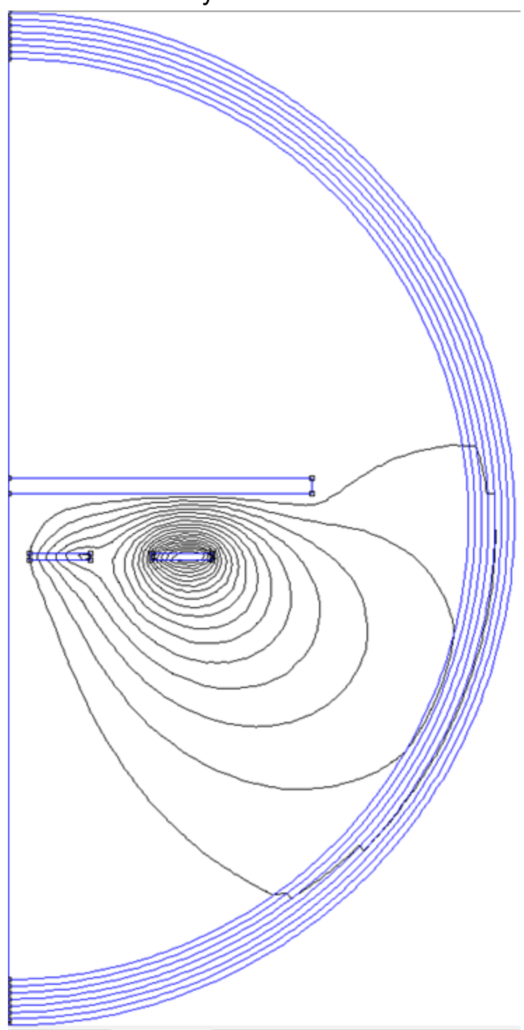
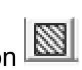


Figure 3-1. Axisymmetric Answer File Showing the Contour Plot of the Magnetic Field Around the Coil and the Target

1. Change the contour plot options
   a. Navigate to **View** → **Contour Plot**, or click on the **Contour Plot** button 
   b. In the **Contour Plot Options**, change the number of contour lines, as well as their upper and lower bounds, and click **OK**.

Copyright © 2021 Texas Instruments Incorporated

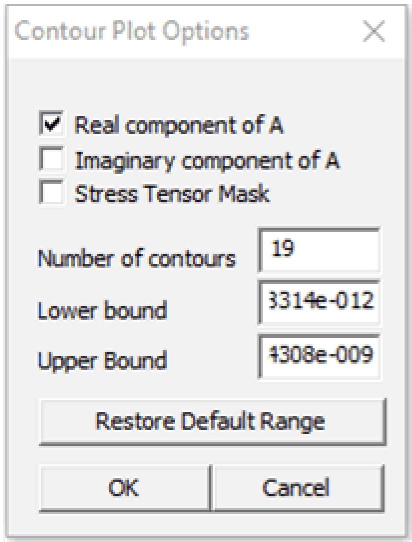


Figure 3-2. Contour Plot Options Menu

2. Add various density plots

   a. Navigate to **View** → **Density Plot**, or click on the **Density Plot** button 
   b. In the **Dialog** menu, check the **Show Density Plot** box.
   c. Select the desired **Plotted Value**, generally |Flux Density| (T) or |Current Density| (MA/m^2). Flux density is useful for determining optimal target placement, because we want the target in the area of highest flux density. Current density is useful for displaying eddy currents.

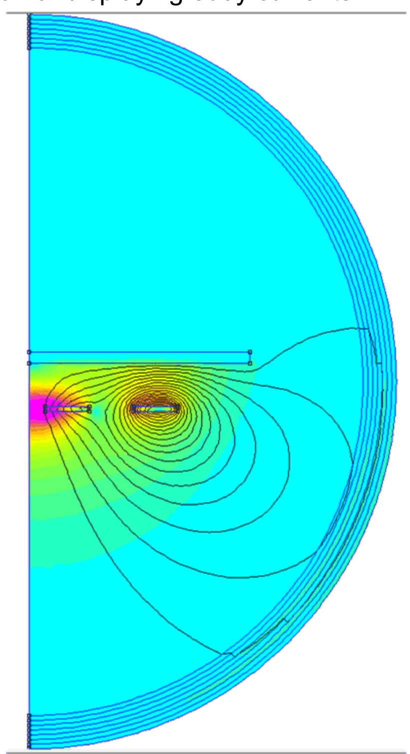


Figure 3-3. Axisymmetric Answer File Showing Flux Density Plot

Copyright © 2021 Texas Instruments Incorporated

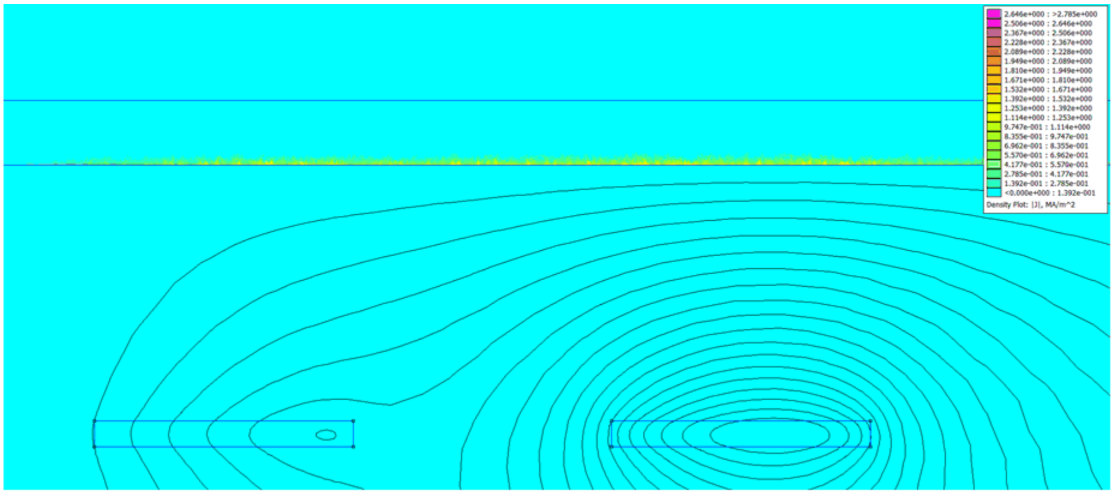
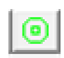


Figure 3-4. Axisymmetric Answer File Showing Current Density Plot That Shows Eddy Currents on the Target

3. Change the mesh

   The mesh is one of the most important aspects in generating an accurate answer file. Generally, it is best to choose a small mesh around the coils, the target, and the axis of symmetry (for axisymmetric problems).

   a. To display the mesh, navigate to **View** → **Show Mesh**, or click the **Show Mesh** button
   b. To change the mesh, return to the .FEM file.
   c. To select the desired material, click the green materials button  then right-click near the desired material node and press the space bar. The **Properties for selected block** menu should open.
   d. Uncheck the **Let Triangle choose Mesh Size** box.
   e. Enter a new Mesh size. A mesh size around 0.01 is a good starting point. Note that the problem must be solved again to reflect any changes.

4. View the inductance of the coil

   a. Click the **Inductor** button  at the top.
   b. The **Circuit Properties** menu should open. The inductance will be listed as Flux/Current.

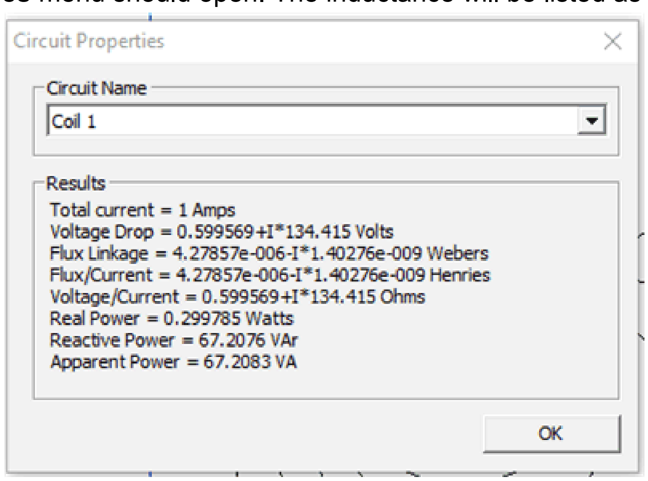


Figure 3-5. Circuit Properties Menu

Copyright © 2021 Texas Instruments Incorporated

4 Calculating Resolution

Most LDC devices measure shifts in inductance by measuring the resonant frequency of an LC tank circuit. For this reason, it is most useful to calculate LDC resolution in units of Hz. However, many users will spec their desired resolution in terms of the distance between a target and the sensor coil, and also want to determine the SNR of the frequency shift associated with a target movement. For these reasons, this section will give a set of steps to use FEMM to simulate the sensor inductance, and use the LDC EVM and software GUI to estimate the SNR.

1. Calculate the Frequency:
   a. A real inductive sensor will consist of the sensor coil and fixed, physical capacitor. FEMM accurately calculates the magnetic fields of the coil - and therefore its inductance - but it will not include the effects of the capacitor.

      To work with this limitation, we describe two sets of steps below. The first will assume the sensor coil geometry and the capacitor value are known, but the coil inductance and tank circuit resonant frequency are unknown. The second set of steps will be be based on a known coil geometry and tank circuit resonant frequency, but the coil inductance and the capacitance are unknown. This will allow you to simulate the coil in FEMM to determine its inductance, and then calculate the capacitance.

      Neither approach considers the effect of the capacitor into the FEMM simulations. Instead, a few calculate/simulate steps will be used to to quickly iterate to the final inductance value and resonant frequency.

      (1) - Known Coil Geometry and Capacitor Value

      i. For the first iteration, pick a frequency and a maximum target distance and complete Part 1. Use the FEMM .ans file to calculate the inductance.
      ii. Use the known capacitance and the newly calculated inductance to calculate the corresponding resonant frequency as shown in Equation 2:

$$f = \frac{1}{2\pi\sqrt{LC}} \tag{2}$$

      iii. Use the new frequency and the original target distance in FEMM to calculate a new inductance.
      iv. Repeat steps ii and iii until the inductance does not change significantly between iterations.
      v. Move the target by the minimum distance for the desired resolution and repeat steps i-iv. Determine the difference in calculated frequencies between the two target positions.

      (2) - Known Coil Geometry and Tank Circuit Resonant Frequency
   b. Complete Part 1 using the resonant frequency and target distance. Use the FEMM .ans file to calculate the inductance.
      i. Calculate the capacitance using:

$$C = \frac{1}{(2\pi f)^2 L} \tag{3}$$

      ii. Move the target by the minimum distance for the desired resolution and perform the steps i - iv in (1) just above. Determine the difference in calculated frequencies between the two target positions.
2. Calculate Resolution Using Simulation Results and Noise Floor Measurements:
   a. Measure the Noise Floor: For inductive sensing applications, the first step is to measure the noise floor for the system. Basic noise floor measurements can be done using the LDC evaluation modules, which can be purchased on ti.com. Accuracy can be improved - with respect to the end application - by making measurements over the expected temperature range, and using a sensor designed for the end application. The default evaluation module, however, will still provide a reasonable noise floor estimate.

   To measure the noise floor using an evaluation module, first download and open the Sensing Solutions GUI. For more detailed installation and use instructions, see the user's guide associated with the EVM.
   a. Open the GUI and navigate to the **Configuration tab**.
   b. Configure the RCOUNT setting so that the LDC device will sample the sensor frequency at the desired rate.

Copyright © 2021 Texas Instruments Incorporated

  - For non-default sensors, the user must use an oscilloscope now to verify that the sensor oscillation amplitude and the sensor frequency are within the acceptable range for the specific device, as specified in the datasheet.
  - If the sensor oscillation amplitude is outside of the recommended range, the sensor drive current will need to be modified. Please see Setting LDC1312/4, LDC1612/4, and LDC1101 Sensor Drive Configuration for instructions.
  - If the sensor oscillation frequency is outside of the frequency range supported by the device, either the inductor or the capacitor in the LC tank will need to be modified.

c. Next, navigate to the **Data Streaming** tab.
d. Click the **Start Streaming** button to begin the noise floor measurement.
e. Ensure that the graph displays **Detected Sensor Frequency (MHz)**, which is controlled by the drop-down menu in the upper left corner.
f. Click on the **Show Statistics** button to display the average and standard deviation. The data's standard deviation will be used to estimate the system noise. Increase the number of decimals of the standard deviation until the last digit is rapidly changing.

Log the data in a .csv file and use your favorite analysis tool (e.g. Excel, MATLAB, etc.) to compute the standard deviation over all variations, including temperature. In general, the noise floor measurements will be more accurate over longer measurement periods.

a. Calculate the Final Resolution via Signal-to-Noise Ratio (SNR):
To calculate the SNR, use Equation 4, using three times the measured standard deviation as the total noise.
  - Remember that the SNR should be at least 10 to achieve the desired resolution in a real-world application.
  - Note that resolution significantly increases when the target is closer to the sensor. The FEMM simulations should test the resolution at both the minimum and maximum target distances using one of the two steps in (1) just above.
  - Also note that very fine resolutions may not be achievable at high sample rates. For more information about how sample rate affects resolution, see the *Optimizing L Measurement Resolution for the LDC161x and LDC1101* application report.
  - If the FEMM simulations show that the desired resolution is not viable, then the sensor design or the target design must be modified. LDC Sensor Design and LDC Target Design are good resources.

$$SNR = \frac{\Delta f}{3\sigma_N} \tag{4}$$

- SNR is the signal-to-noise ratio of the inductive sensing system
- Δf is the calculated frequency shift from 1
- σ_N is the measured standard deviation of the noise from 2.a

5 Other Resources

For inductive sensing simulations, the Excel_FEMM tool can be especially helpful to speed up the simulation process.

For basic target shapes, the tool can automate the simulation process and will output the simulation results. It can also be useful to automate the creation of the coil in FEMM, and the resulting .fem file can be modified for custom target shapes. This tool is included with the LDC Tools Excel calculator.

For more general instructions and help using FEMM, please see the Finite Element Method Magnetics : Homepage.

6 Summary

This application note provides the step-by-step instructions for simulating inductive sensors using FEMM. The simulation results can be especially helpful to estimate system resolution and determine whether sensor design changes are needed before creating a prototype. Because the simulations are approximations of the real sensors, some differences between the real prototype and the simulation results are expected.

Copyright © 2021 Texas Instruments Incorporated

7 References

- D. C. Meeker, Finite Element Method Magnetics, Version 4.2 (28 Feb 2018 Build), http://www.femm.info

8 Revision History

NOTE: Page numbers for previous revisions may differ from page numbers in the current version.

Copyright © 2021 Texas Instruments Incorporated

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATA SHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES “AS IS” AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, regulatory or other requirements.

These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI’s products are provided subject to TI’s Terms of Sale or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI’s provision of these resources does not expand or otherwise alter TI’s applicable warranties or warranty disclaimers for TI products.

TI objects to and rejects any additional or different terms you may have proposed.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2022, Texas Instruments Incorporated