

Simulations and Tests for Dual Helicity Targets

And an introduction to COMSOL Multiphysics

Josh Pond,¹ Michael Worcester,¹ and Dustin Keller^{1, a)}
University of Virginia Physics Department^{b)}

(Dated: 21 December 2016)

Some preliminary modeling and testing of the microwave cavity (cryostat geometry), DC-holding coils, and NMR coils are presented for use with the CLAS12 horizontal fridge for longitudinal solid polarized targets. An introduction to COMSOL Multiphysics is also provided for use with the microwave module simulator. COMSOL Multiphysics is a graphical simulation suite that is capable of simulating numerous physical situations on a geometric model that can either be created within COMSOL itself or imported from most CAD programs. The simulation described is to investigate various cavity geometries for irradiating two target cups with a single microwave input to achieve positive and negative polarization simultaneously.

The longitudinally polarized target program in Hall-B seeks to incorporate a target system which can polarize positively and negatively using two separate polarized target cells in the beam-line simultaneously. The drawback of the traditional single cell target system is that significant time dependent errors acquired from changes in systematics during data collection at two separate times for each target helicity leads to false asymmetries. This type of uncertainty limits the scale that can be probed for small asymmetries. Significant reduction of this type of uncertainty can be achieved by having two targets polarized in opposite directions in the beam-line to collect data in both helicity states at the same time.

One of the critical challenges which still must be addressed is how best to irradiate two cells at the same time with one source so that the microwave power each cell receives is similar and maximized. This will allow both cells to be ready for beam on a similar time scale as well as reach and maintain an optimal polarization level through the course of the experiment. It is also necessary to study coils that can change the holding field for each target cell independently so that the microwave resonant frequency need not vary in order to polarize opposing target helicities. Looking into NMR coil optimization for this dual cell configuration for polarization measurements is also beneficial.

COMSOL Multiphysics has been used to simulate the nose cavity which can later guide prototyping and future work on a physical model. This note starts with an introduction to the use of COMSOL in Section 1. An overview of the process of testing some geometries of the target nose piece using the microwave module is given in Section 2. In Section 3, the DC-field module is also used to discuss the coils used to hold each cell at the field appropriate for positive and negative polarization. Finally, in Section 4, some NMR coil tests are reported from a recent cool down at UVA.

I. COMSOL MULTIPHYSICS

A. Overview

COMSOL Multiphysics is a powerful simulation environment software suite. It is available on Windows, Mac, and Linux and is available with several extra modules for added capabilities. This simulation was done with COMSOL 5.1 with the additional Radio Frequency module on a Dell Precision Tower T5810 with an Intel Xeon CPU E5-1660 v3 running Windows 7 Enterprise.

B. Start-Up

After starting the program and after the splash screen the first thing that appears is the option for either a blank model or using the model wizard. Choose the model wizard option. The first option within the Wizard is the space dimensions and the most likely choice would be to choose three dimensions.

The next stage will be to choose the physics and the study of the model. The physics will define the broad category of physics that will be simulated on the model. The study is how the simulation will be carried out and reported and serves to narrow down the scope of the simulation. Choose Done and the complete COMSOL GUI will appear with a blank graphics window.

C. User Interface

There is too much contained in the COMSOL user interface to describe entirely and in great detail, but only a rudimentary understanding is necessary to begin working on a simulation. Here are some highlights.

1. Ribbon

The Ribbon is the large toolbar at the top of the interface common to Microsoft programs in the past several years, and the functionality is the same. There is a file

^{a)}Electronic mail: jpond@jlab.org, mw9va@virginia.edu, dustin@jlab.org

^{b)}<http://www.phys.virginia.edu/>

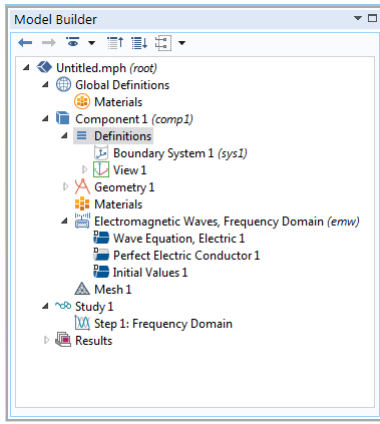


FIG. 1. Model Builder Panel.

button with all the standard save, open, and preferences menus one might expect. The Ribbon also has tabs for all steps of the simulation process that collect all the functions that fall under these categories. All parts of the process of building the model, simulating, and post-processing can be initiated from the Ribbon.

2. Model Builder Panel

Most of the steps of the model building process can be handled in the aptly named Model Builder Panel which functions as a step tree from top to bottom of the steps from defining parameters to post-processing. Each entry can be right clicked to obtain the rest of the menus of functions that can be performed on the object. Following the step tree all the way down is the best way of staying organized during the simulation process.

3. Settings Panel

The Settings Panel is the most important panel in the interface. Almost every action a user can take on an object will be carried out in the Settings Panel. Click to add a new geometric object and the parameters of the object are defined in the Settings Panel. Add a new material to a domain and the material's preferences are defined in the settings panel. It is a dynamic interface and one of the main points of contact for the user.

Example: When creating a new geometric object like a sphere, as is the case in figures 2 and 3, the Settings Panel is where everything about the object is input.

4. Graphics Window

The Graphics Window is where the model and any model dependent 3-D plots will be displayed. Left-click and drag for traditional center point spherical rotation

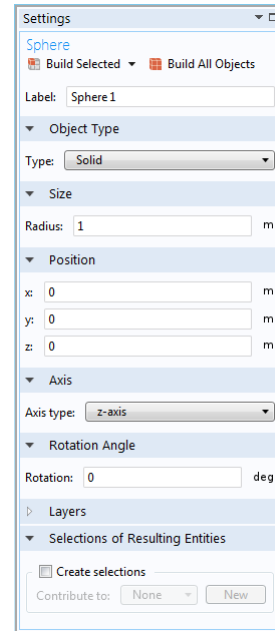


FIG. 2. Settings Panel.

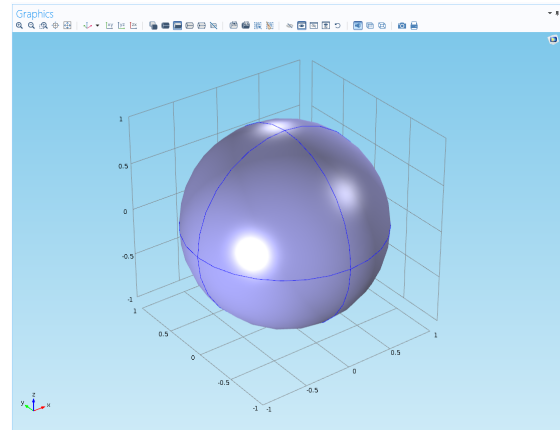


FIG. 3. Graphics Window.

and right-click and drag for sliding. The graphics toolbar has functionality for making the model transparent and a wireframe, as well as hiding any object of any level. It also has quick orientation selection buttons, normal zoom functions, screenshot, and print buttons.

The Graphics Window is also where the user does selections of objects in the model for settings and manipulations.

5. Other Interface Aspects

The COMSOL interface has, by default, two other panels that hold several random tabs, including among other things, a log, system messages, progress statistics, and the material selection list. These tabs, as well as any

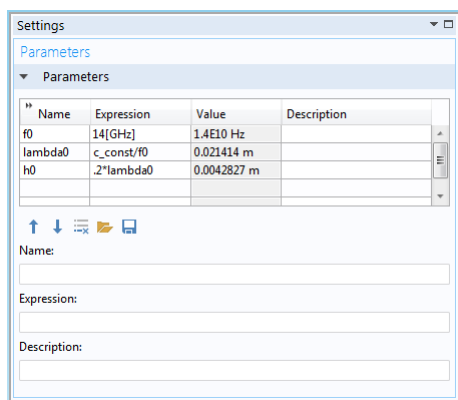


FIG. 4. Radio Frequency Parameters.

other panels described except for the Ribbon, are free to be undocked, moved, re-docked, resized, or left floating anywhere on the screen. The GUI layout is almost entirely up to the user.

The Help button is in the top right-hand corner and will display a help panel with information on whatever is currently selected in the interface. For example the user can click on the Add-Study tab and click the Help Button and the entry on the Add-Study tab will be displayed.

D. Building a Model

The part of the simulation process that requires the most from the user is the model building process. This introduction is assuming the user will be creating the geometry of the model in COMSOL using COMSOL primitives. It is possible to import the geometry from a CAD file, and that would be recommended for intricate designs. A word of caution to note early is that the more powerful CAD import tools are not included in the base COMSOL package, and will cost extra to get the most out of COMSOL and CAD. The included base import tools cannot be recommended at this time to be relied on.

1. Parameters

In the home tab on the Ribbon, under the sub-section definitions, is a button labeled Parameters. When it is pressed the parameters menu appears in the Settings Panel. It is recommended that the user store any values that will be used throughout the simulation here under a named variable. This might include frequencies, or magnetic fields, or some other experiment specific value. A clear case of this function's usefulness will be provided in section 3.

Example: Figure 4 shows the recommended parameters for simulating an EM wave traveling through air. The purpose of h0 will be clear soon.

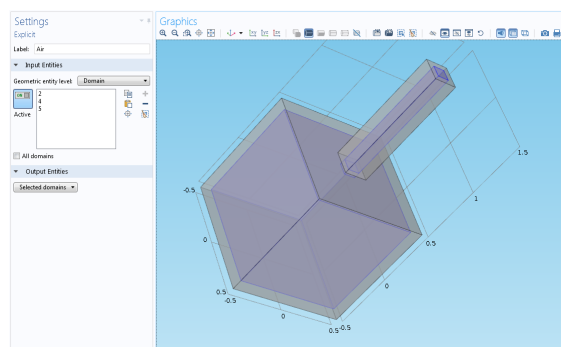


FIG. 5. Explicit Selections.

2. Geometry

The first real step of the model building process is defining the geometry of the model itself. The model geometry is comprised of geometric primitives that can be selected from the geometry tab in the Ribbon, or by right-clicking the geometry entry in the Model Builder. By clicking on the Geometry step, though, the user can also get access to several settings that will pertain to the geometry as a whole, such as the scale and the units.

The model is fractured into different selection levels: Object, Domain, Boundary, Edge, and Point, which can individually be selected in the Graphics Window by choosing the selection level in the Window toolbar and clicking on it in the Window itself. Domains are by far the most important level. For example, when building a model the user should always make the primitives solid, and create cavities by overlaying a larger primitive on top of another smaller one. The software will create two domains with the inner primitive as one and the excess as another, and as will be clear in the next section, this is enough to make each domain a different material and create a physical cavity.

The actual act of building a single primitive is remarkably easy. As can be seen in Figure 2, the user simply chooses the primitive from the Ribbon, the dimensions and the position in the Settings Panel, and then clicks Build All.

It is highly recommended that once the entire geometry is built to use the Explicit definition function, found under the definition tab on the Ribbon in the Selections sub-section, to define a group of domains under a named variable. This does not actually combine the domains into a single domain, but in the future the user can select this variable from a menu instead of selecting domains by hand and the software will select all included domains.

3. Materials

Materials are among the most important parts of the simulation process. They're how COMSOL knows how to solve the study and defines much of the physics. This

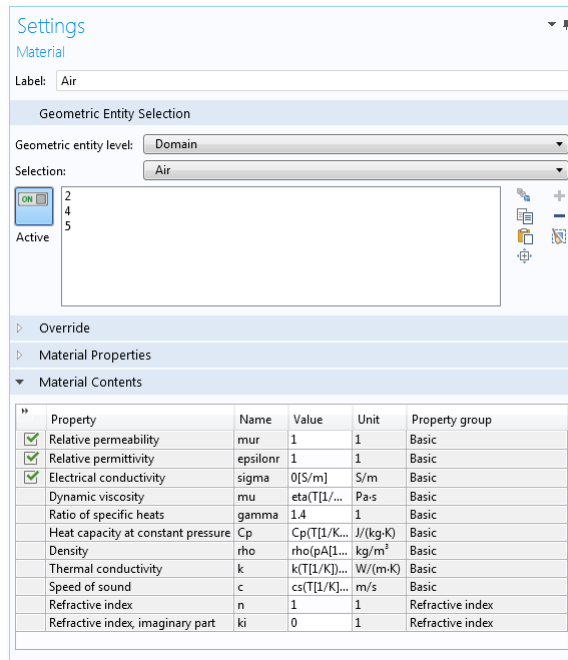


FIG. 6. Materials.

is where the explicit selections come in handy. To add a material to the model, click Add Material either from the Ribbon, or by right-clicking Materials in the Model Builder Panel. The list of materials will appear in one of the miscellaneous panels, by default on the right side of the GUI. Double-Click the material and it will be added to the Model Builder Tree. The user can choose any number of materials at this step. Click on the Tree entry for that material and its settings will appear in the Settings Panel. The drop menus in the Geometric entity level (Selection Level) area are, by default, set to Domain and Manual respectively. Change manual to the explicit variable name that was chosen earlier and those domains will be added to that material.

4. Applied Physics

This is the most important step in the process, but depending on what physics was chosen all the way back in the Model Wizard, it can be the easiest. The way it is set up is entirely dependent on that choice, so a perfect introduction to each possible physical simulation is impossible. So an example of the RF Microwave module will be given. Some overall recommendations, in no particular order, would be:

- The user decides which domains are in the scope of the physics, so only choose the domains where it is meaningful.
- The user will most likely not need to change any default settings added to the Model Tree by the Model Wizard.

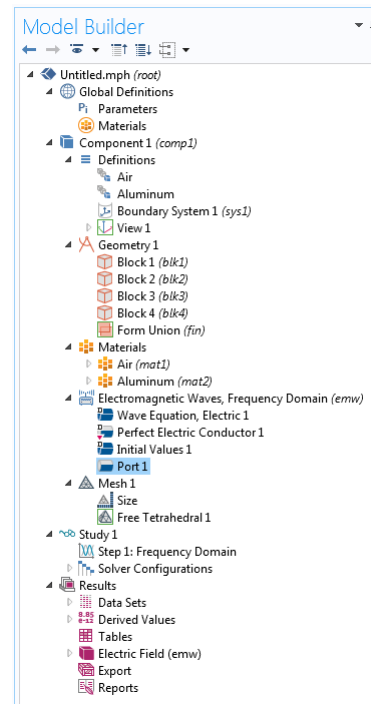


FIG. 7. MW Example Model Viewer.

- The user may need to add boundary conditions, or some other physical attribute to the model, depending on the particular physics being studied.

Figure 7 shows the Model Builder for this example. The Physics entry, Electromagnetic Waves, Frequency Domain (emw), as well as the branches, Wave Equations, Perfect Electric Conductor, and Initial Values are left the same as added by the Wizard, but a Port is added by selecting Port from the Boundaries menu under the Physics tab on the Ribbon. The Physics is only applied to the Air domains in the Air explicit definition made earlier, because the penetration into Aluminum is so shallow that studying it would not be meaningful. By clicking on the Physics Tree entry, Electromagnetic Waves, Frequency Domain (emw) the user gets the Settings Panel for the physics model as a whole. For this example the only change needing to be made is switching the Domain Selection from All Domains to the explicit named definition Air that was created in the geometry step. The next consideration is the Port1 entry.

The Boundary selection defines the physical location of the port. The Type of port is dependent on its shape. There are built in types like circular, coaxial, and periodic. For this to be an input port, the Wave excitation needs to be On. The Mode type and Mode Number should be fine as default, but the documentation can be consulted. (If this port were on a boundary between two domains that are in the physics scope, the Activate slit conditions box will need to be checked.) Any number of Ports can be added to a model, and leaving Wave excitation off will make them exclusive output ports. The

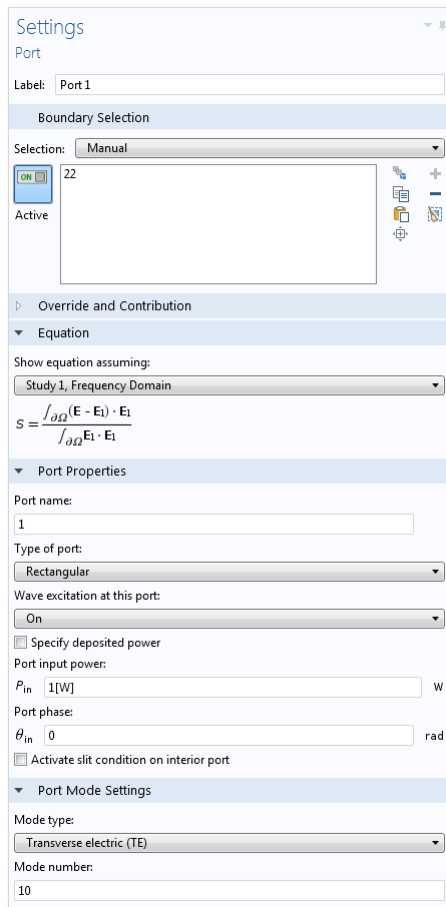


FIG. 8. Port Settings.

Port is chosen to be at the top of the waveguide by clicking that boundary in the Graphics Window. So the final model appears as figure 9.

5. Mesh

The model is ready to mesh at this point. The mesh defines the lattice that the physics equations are solved on for the study. Much like the Physics portion, this will be dependent on the particular model, but less so. Most users should be able to make due with a Free Tetrahedral mesh. Right-click on the Mesh 1 Tree entry in the Model Builder and click Size. This will add a Size entry to the Model Tree and its settings should appear in the Settings Panel. For most users the default settings, which define the mesh sizes tailored to the physics, will be fine. In the example a special case will be shown. Next, click Free Tetrahedral from the Mesh tab on the Ribbon. This will add the Free Tetrahedral entry to the Model Tree and its settings should appear in the Settings Panel. The only setting that should need to be changed for most users is to switch the Domain Selection to the same domains that are in the Physics Scope. Click Build All and the mesh

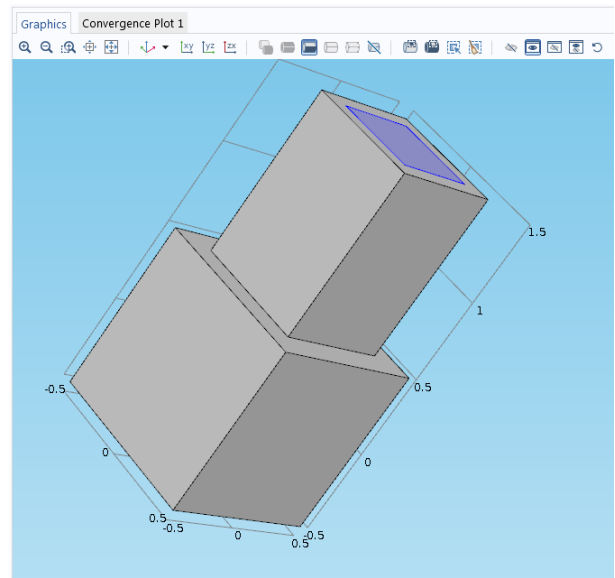


FIG. 9. Example Completed Model.

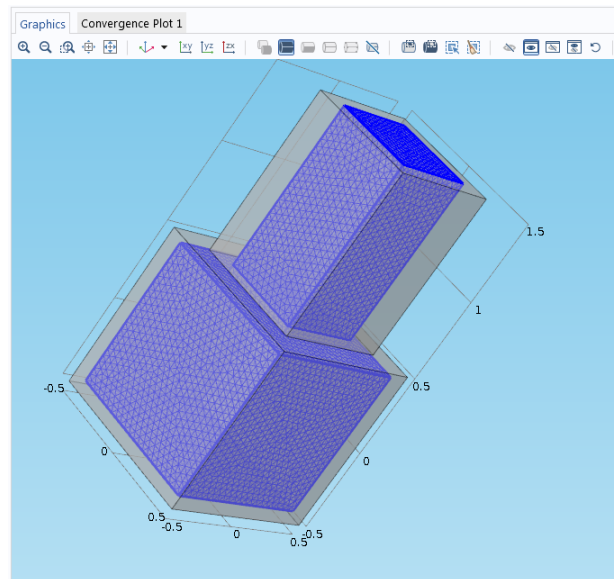


FIG. 10. Mesh.

should appear in the Graphics Window as figure 10.

For this particular mesh, the Element Size has been customized to accommodate the fact that in an RF Microwave study the user would want the element size to be smaller than the wavelength of the microwave itself. So the default Size settings are replaced with what can be found in figure 11.

E. Study

The study is the heart of the COMSOL software. There are numerous different studies to perform and

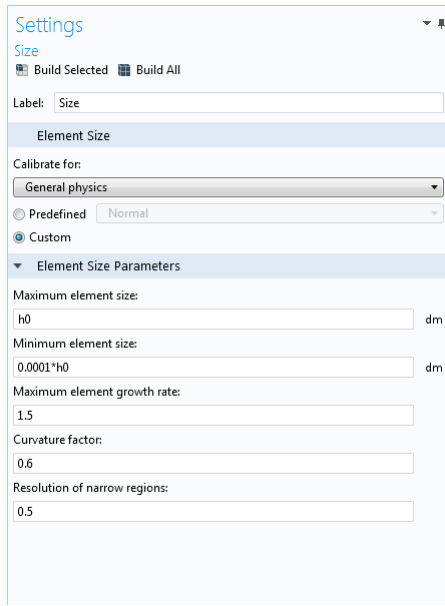


FIG. 11. Custom Mesh Element Settings.

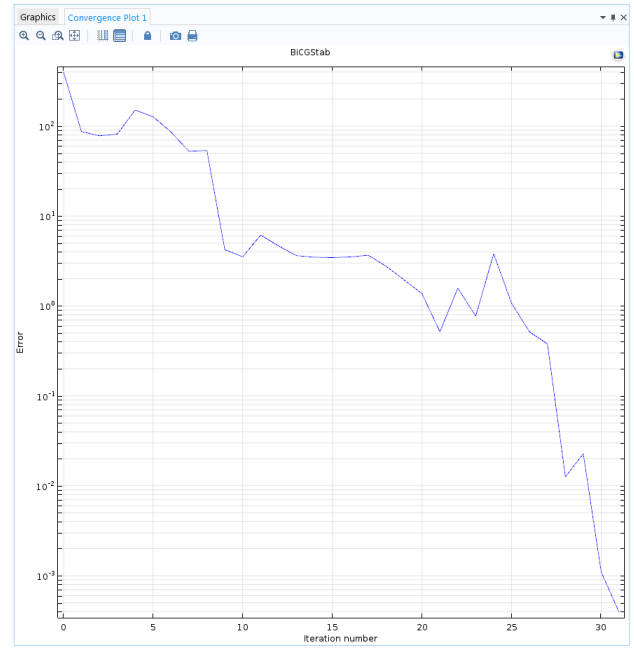


FIG. 12. Convergence Plot.

they are all available through the Model Wizard, or can be added later through the Ribbon tab, or the Model Builder. Once again most users will be satisfied with the study the Wizard assigns. The study is performed in a series of steps that are represented in the Model Builder Tree. Many seemingly simple models will need multiple steps. For example, a Port shape not represented in the Port Settings will need to be solved numerically and stored for the next step of the study. The documentation should be consulted on these matters. The only general tip would be to check the settings for each step by clicking on their Tree entry. Most will have one or two settings that will need to be set by hand. For example in our Microwave model, the input frequency Units and Frequencies need to be set by hand. For this model they are just GHz, and the f_0 parameter which is in gigahertz. The default Solver configuration Tree entry will be fine for most users. For our simple RF Microwave model, there is only one step, the Frequency Domain step. After adding the frequency and clicking = Compute a Convergence Plot 1 tab will appear in the Graphics Window when the study progress gets to 70%.

When the Solver, which in this case is a Stationary Solver, completes, the study will be complete.

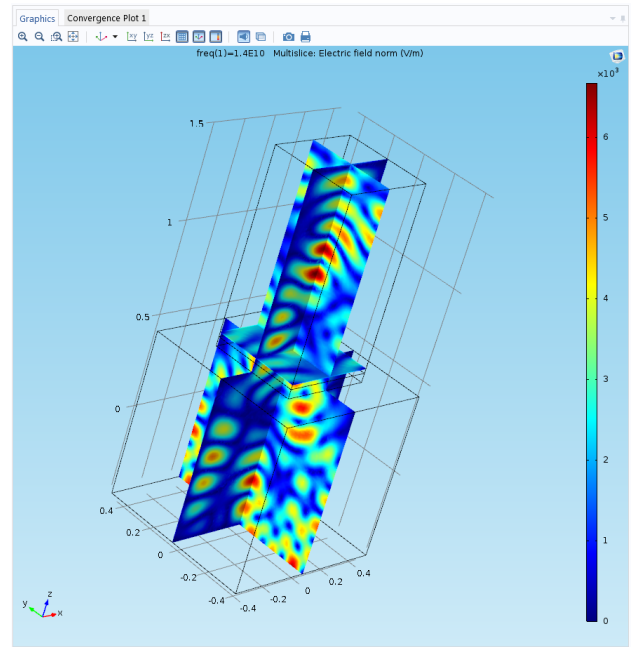


FIG. 13. Multislice Plot.

F. Post-Processing

Post processing is where COMSOL Multiphysics really shines. Once the Study is finished the default Multislice plot (figure 13) will appear in the Graphics Window, and its Tree entry will appear under Results in the Model Builder.

1. Plots

COMSOL has many plots to choose from to display any number of physical values graphically on the model. Some of the most visually dynamic plots, especially for the RF Microwave study represented here, include the contour plot (figure 14) and the arrow volume plot (figure 15).

These plots can be found by right-clicking on the Tree

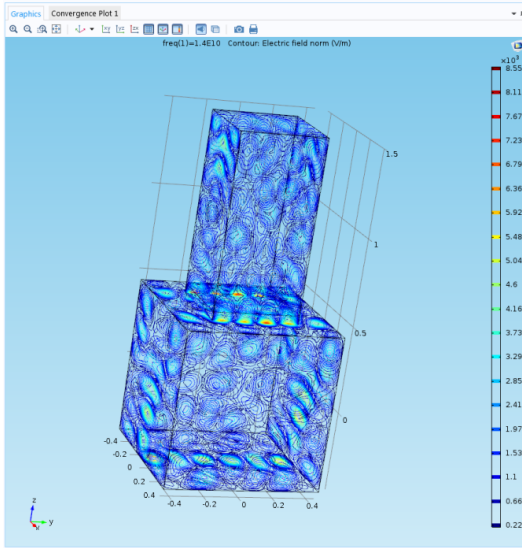


FIG. 14. Contour Plot.

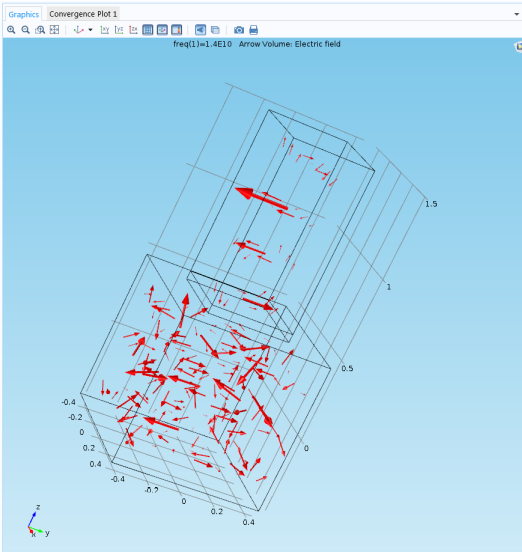


FIG. 15. Arrow Volume Plot.

entry under Results which corresponds to the Physics and selecting the plot name, or by going through the Results tab on the Ribbon. The settings for that plot will appear in the Settings panel, where different parameters can be altered to the user's desires, and pressing Plot will overlay the plot on the model. Multiple plots can be overlain and they can be interchanged by right-clicking on the Tree entry and choosing Disable / Enable as the situation demands.

2. Derived Values

COMSOL can evaluate untold numbers of expressions using built in variables calculated in the study. These

can be found by right-clicking on the Tree entry Derived Values and selecting the Value name, or by going through the Results tab on the Ribbon.

For example, to get a surface integration of the norm of the power across all interior boundaries in our Microwave Model, the user would start by defining an explicit definition, just as above, choosing the same domains as in Air, but changing the Output Entities from Selected Domains to Adjacent Boundaries and checking the exterior boundaries check-box. Next right-click on the Derived Values Tree entry and select Integration, then Surface Integration. The settings for the surface integration will appear in the Settings panel. Select that definition's name from the drop menu.

For this value there is no built in variable for the norm of the power, but the user can construct this value with the built in component variables, with the form $(emw.Poavx^2 + emw.Poavy^2 + emw.Poavz^2)^{.5}$. These different built in variables can be found by clicking on the Green and Red compass needle button in the Settings panel for the Surface Integration.

When the expression is complete click compute and the value will be stored in a table, and will appear in the tab with the table's name in the panel that is, by default, under the Graphics Window. Any number of computations can be done with the same Derived Value Tree entry and they will be added to the same table, and any other Tree entries will be given their own table. Whichever Derived Value Tree entry is selected will determine which table is visible in the tab.

G. Concerns and Limitations

COMSOL is not a perfect software package for all studies and there are several limitations to be considered.

- COMSOL uses an immense amount of memory to solve the linear equations, and the smaller the mesh and the more elements there are, the more it will need. The software uses a Virtual Memory Stack, provided by the operating system, where it takes memory from different physical locations to create a continuous block of memory which the software sees as one device. Even with this tool memory usage can sky-rocket fast, even 64 gigabytes of RAM will begin to feel small if simulating models with hundreds of thousands of degrees of freedom.
- The materials list is expansive, but will become limited quickly in many specialized fields.
- Computation times for the Study step can exceed several hours for models that seem small and simple to calculate.
- As stated previously, the CAD import capabilities of the base software range from limited to outright unusable for most models.

- The online community is limited and many problems that appear simple will not be addressed in the forums. The software comes with live email support from COMSOL engineers included who can be incredibly helpful, but turn around can exceed 24 hours for involved questions.

II. DUAL HELICITY NOSE

One of the goals of using COMSOL Multiphysics will be to engineer an optimal nose cavity for irradiating two separate target material cups with a single microwave input to assist in the building any physical prototypes. This is, more than anything, a case study in the effectiveness of computer simulation prototyping using the microwave module.

A. Set-up

The set-up for our model is a polarized target experiment in which two separate targets are polarized opposite of each other and struck by the same beam. The two cups will be coaxially aligned in the beamline, within the same nose cavity and will interact with the same beam. The different polarizations will be initiated by a large background magnetic field that will be tuned by a small local magnetic coil. If we irradiate the cups with the same microwave source we increase efficiency and increase our ability to make the targets as similar as possible, optimizing their polarizations, and maximizing the target figure of merit over the course of the experiment. The goal of the project is to show explicitly that there is an optimal geometry of this nose and the microwave delivery that will maximize irradiation seen at each cup to a similar magnitude.

B. The Model

The model began life as a physical prototype that was mimicked with COMSOL primitives as a control and benchmarked for further prototyping (Figure 16).

The microwaves are guided into the cavity through the circular waveguide that enters through the top of the nose. (quantitative results for each cup before modification?) The staggered wedge redirects them towards the area where the cups would be. The aluminum outer skin of this model is removed, so only the Air cavity is shown and the plastic cups are removed. This model was studied using the RF Microwave module just like the simple example in section 2 and measurements of intensity were taken and compared to the physical prototype.

From the basic premise different models were designed and prototyped as a simulation. First the structure at the end of the waveguide was eliminated and the geometry of the cavity itself was used to guide the microwave.

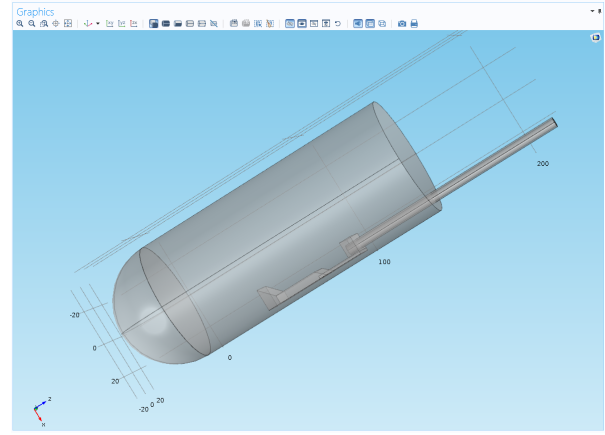


FIG. 16. Benchmark Nose.

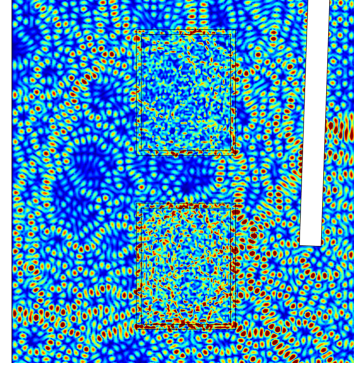


FIG. 17. Close up of the cups with Mica filling.

Then different endcaps were tested, such as flat disks and smaller hemispheres. Staggered cavity walls were tested and multiple positions of the wave guide have been simulated, but they are all primarily an aluminum cavity with a circular waveguide and two cups. (quantitative results of endcap study?)

Another variable that was simulated was the absorption properties of the material in the cups. At resonance the target material should act as an absorber, it has not yet been tested as to how much of an absorber it is. Obtaining imperial information via a cooldown will be a necessary step so that the target material properties can be accurately represent to better simulate how much microwave passes though both cells at resonance. For now, different materials were tested to find one that could be used for a stand in until this information can be obtained. Mica was found to be a good approximation for a target material which still allows some ($\sim 10\%$) microwave at 140 GHz to pass through. The absorption in the cups can be seen in figure 17. (This is where the demo for the absorber test goes as well, Micheal's figure.)

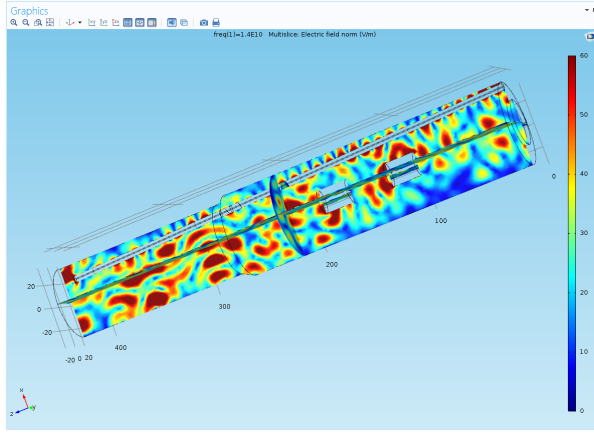


FIG. 18. Prototype Nose.

C. Simulation

The simulation was performed just as the example in section 2. The Microwaves were input by a port on the end of the waveguide and the input frequency (f_0) was set to 1.4 GHz. This is an order of magnitude small than what is required but at this frequency it is possible to fully mesh and run in 3D with the hardware available. Initial simulations in 3D give a qualitative depiction of the microwave behavior in a certain cavity geometry. However for realistic quantitative analysis it is necessary to study the geometry in 2D slices so that COMSOL can handle the higher frequencies. The memory usage limitation described in section 2.7 was the major barrier for high frequency in 3D. The real experimental frequency of interest is around 140 GHz, but at this frequency the wavelength was so small that COMSOL could not make a 3D mesh small enough for a good simulation.

1. Metrics

The metric by which a good or bad geometry was measured was the ratio of the surface integral of the norm of the power through each cup. A surface integral, like the one described in section 2.F.2, was taken over each cup individually and the ratio was calculated such that the closer to one the more even the irradiation is to both cups. The actual magnitude of the power at each cup was also an important factor. (this metric should now be specific to only 2D)

2. 2D Simulation

The best way to get more information for our simulation is running the model in 2D, an arbitrary number of slices can be studied in 2D to get a vary good estimate of the microwave behavior in a 3D structure that could not otherwise be simulated on a realistic time frame. By

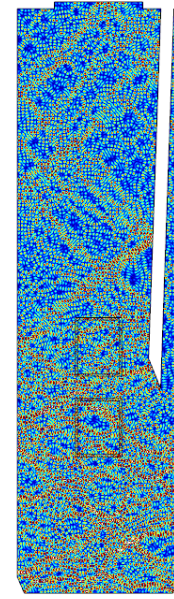


FIG. 19. 2D Simulation. (We need a description of the aspect of what are simulated, if KelF in there, where is the cups and what are they made of?)

decreasing the total number of mesh elements so considerably, we were able to run at 140 GHz. Slicing a 3D structure to make sure to study all aspects and interference of the cavity can be incredible laborious and time consuming so at this stage we primarily focus on a single slice of the full cavity along the center of the waveguide. Additional information can be acquired once some geometries are ruled out using this simpler projection.

Figures 21 and 22 show the variability of the 2D simulation process, and how much control over the geometry of the slice this method gives the user. Figure 21 in particular is representative of how much the cavity geometry affects the way the microwaves propagate throughout the space. (A description and picture of the JLab two celled geometry is required, this is what we ultimately used as the dimensions for our final tests. We should have their pictures included and then our 3D and 2D models of those dimensions.)

D. Results and Current Status

Figures 20 and 22 show the most recent simulations as of the time of writing. Working from a computer drawing, a new geometry was created in COMSOL. The recreation was designed to closely fit the physical specifications of the original drawing as well as be composed of similar materials that were intended for the design. The rounded nose has thus far given the best results, by irradiating the cups the most evenly. (We want irradiation in both cells similar and high, and need the numbers, a table for each geometry variation. The ultimate goal here

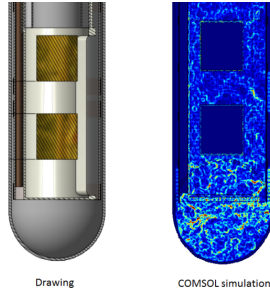


FIG. 20. Side-by-side Comparison of Nose Drawing and COMSOL Simulation.

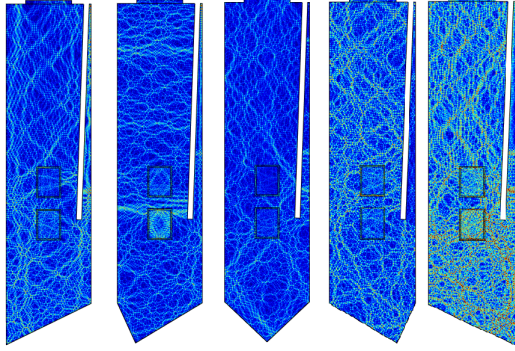


FIG. 21. Varying the point position of a conical cap.

is to introduce a best possible nose piece with dimensions as a preliminary study that can be varied later in necessary. We need much more emphasis on dimensions and quantitative results.)

III. DC HOLDING COILS

The objective of this study is to investigate how best to induce polarization in each of the target cell for two distinct resonant frequencies by changing holding field rather than microwave frequency. The two holding coils were also modeled in COMSOL. The purpose of each coil system is to create a smaller additive or subtractive magnetic field inside of the 5 T background field. These smaller fields would surround each target so as to create

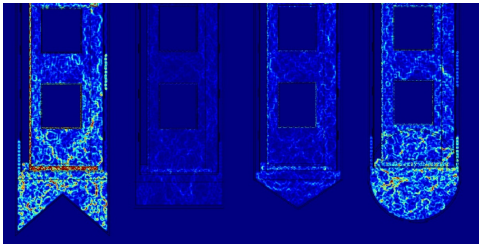


FIG. 22. Various Nose Geometries.

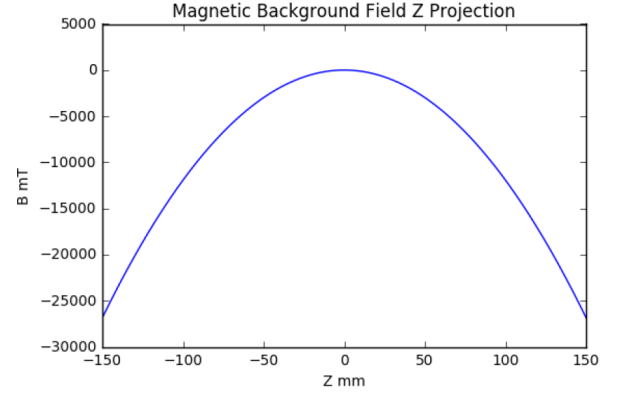


FIG. 23. Z Projection.

the correct environment to induce resonance in the target material using a single microwave frequency produced at the waveguide opening.

A. The Background Field

Fortunately, COMSOL allows for simulation of a background magnetic field so long as a polynomial expression as a function of position is provided. The field map used in the CLAS12 Geant simulations of the detector package was analysed and fit to acquire a field function for the background holding field. The strength of the field in the target region is on the order of 5 T. The polynomial expression from the fit used to simulate the background field was,

$$B_z = 5 + (2.249e-12)y + (2.538e-14)z + (0.6778)y^2 - (8.184e-13)yz - (1.193)z^2 - (8.797e-9)y^3 - (1.202e-10)y^2z - (9.568e-12)yz^2 - (2.94e-12)z^3$$

To input the expression, the equation was pasted into Components, then Magnetic Fields, then Settings, then Background Magnetic Field. The background has the form of Figure 23 along the beam-line in the region of the target cups.

B. Individual Coil Systems

For the sake of field uniformity, each coil system was modeled with the Helmholtz configuration. The Helmholtz configuration was a convenient choice because it is already optimized for peak uniformity. The coils were created by using Geometry, then More Primitives, then Helix. Each coil was given a Major Radius of 2.59 cm and a Minor Radius of 0.01 cm and an Axial Pitch of 0.05 cm. Additive coils (that is, coils that would add to the background magnetic field) were given a Right Handed Chirality, while subtractive coils were given a Left Handed Chirality.

Due to meshing limitations, the coils could not be wound so that there would be no empty space between

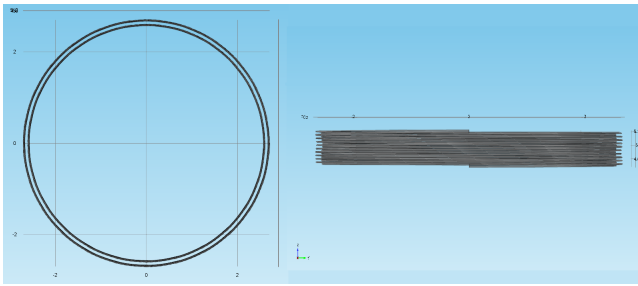


FIG. 24. Two Coils Act As One Airtight Coil.

current loops. To circumvent this issue, duplicate coils of each coil were created with slightly larger Major Radii and a small translation along the z axis. See Figure 24.

C. Adding Physics

To simulate a current running through the coil pairs, the Magnetic Fields (mf) Physics was used. Virtual coils were created by right clicking Magnetic Fields (mf) under Component, and then selecting Multi-Turn Coil toward the top. The virtual coil was then assigned to a specific geometry by selecting Multi-Turn Coil 1 and then left clicking the desired geometry. To give the program an understanding of current direction, click the drop box next to Multi Turn Coil 1, then select Coil Geometry 1, and then select a current path along the geometry. In order to reach a difference in frequency of around 140.150-140.700 GHz a range no greater than about 15 mT would be required. We can achieve this by applying .52 A in one coil and 0.491 in the other coil in the opposite direction. (Plot of table with field strength as a function of current.)

D. Materials

COMSOL is unfortunately quite limited in the stock material options in terms of modeling superconductors. Since the default materials library lacked the required alloy (Nb-Ti), a makeshift solution was created by modifying the material properties of titanium. (How was it modified, what did we change we need all the specifics here. What is the argument that our modifications are reasonable in regards to matching Nb-Ti?)

E. Measuring Magnetic Field

COMSOL offers an extensive library of tools for measuring observables. To measure the magnetic field inside each coil system, a two step process was used:

1. Create an array of entities where each entity has an exact position in the domain.

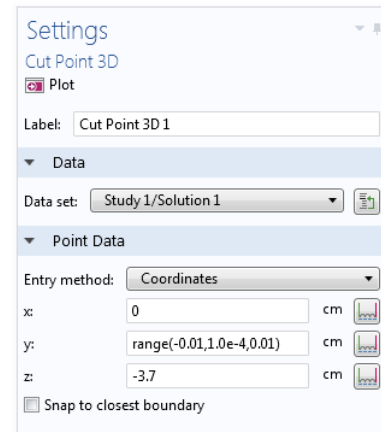


FIG. 25. Cut Point 3D Settings Window.

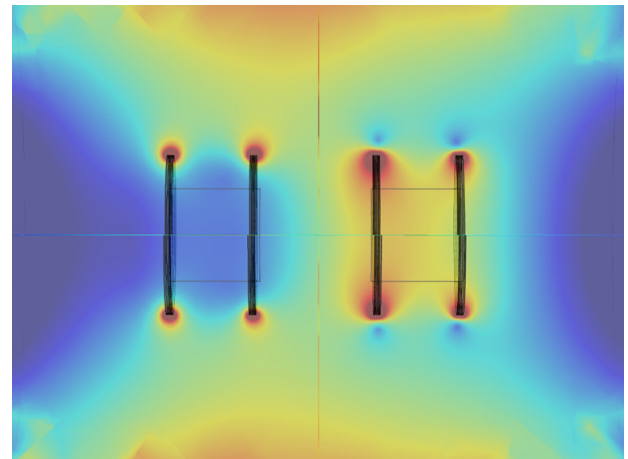


FIG. 26. The Helmholtz Pairs, the additive pair is on the right while the subtractive pair is on the left.

2. Evaluate the magnetic field at each entity position.

To do this, the Cut Point 3D function was used to create an array of entities. The function is located under Results, then Data Sets. See Figure 25.

To evaluate the magnetic field at each entity position, the Point Evaluation tool was used. Point evaluation is located under Results, then Derived Values.

This specific process measures the magnetic field at 200 points radially with a fixed z position. (We need a line projection along Z after the coils have been added, a few would be good at different currents, and a table with numbers of the results of this study.)

IV. NMR TESTING

With possible improvements on the horizon to the traditional Liverpool Q-meter design it is important to have a functional standard for comparison. The most critical aspects of the Liverpool or other constant current designs with similar phase characteristics are the signal to

noise, the linearity of the system, temperature sensitivity of the electronics, and RF-power at the NMR coil. While temperature sensitivity can usually be handled with external regulation the linearity of the system is inherent in the design and an essential component of maintaining effective calibration. The linearity can be studied by taking many measurements of the modulator output over the variation of the input pertaining to polarization changes. This can be studied over the relevant frequency domain using the diode output. It is also necessary to study this with phase dependence over the modulator output. RF-power at the coil can be difficult to measure directly but it is possible to calculate and measure the spin state population deviation using a sensitive low cooling power cryogenic system and a TE signal. In design, minimization of the spin flips induced by the NMR is usually approached by trying to minimize the current while not diminishing the overall signal to noise ratio for the spin species of interest. The coil current with the standard operating conditions is around 0.2 mA for the traditional Liverpool system. Here we show some results on the signal to noise of this system for comparison in future work.

To study signal to noise the Q-meter and Yale card are setup in the standard configuration and data is taken both inside and outside of a cryostat at 1 K and 5 T. Systemic electronic noise can manifest from many sources but regular sinusoidal like patterns are indicative of leaky electronics or bad grounding. Its essential to have a common ground for the NMR and the cryostat. Assuming the NMR has been setup for optimal performance with the surrounding RF-ambiance and instrumentation the signal to noise can be studied by using a crystal resonator positioned at a fixed distance from the coil of the tuned NMR.

Data is averaged over 20 sweeps of 500 scan steps with 0.8kHz per step with a modulated width of 400 kHz and a sample rate of each step of 15287Hz. The sweep time up plus down is 65.4 ms. We use a hardware trigger in the system to avoid software delays. The metric used to study the signal to noise is,

$$SNR(\omega_i) = \frac{1}{\sqrt{N}} |\bar{V}(\omega_i)| : \sigma_n(\omega_i) \quad (1)$$

where \bar{V} is the mean of the signal for a frequency ω_i . There are $i = 500$ central frequencies, one for each scan step. The standard deviation σ_n in \bar{V} is a measure of the noise at i . N is the number of sweeps. To study the absolute noise level, σ_n is also taken with no polarization or crystal signal. In each case a proton like coil of approximately 1.5 cm in diameter is used with the NMR tuned to the proton Larmor frequency.

For the study outside of the cryostat the greatest amplitude of the crystal resonator signal is chosen and around 100 measurements are taken. The mean of the signal was -1.98×10^{-2} (9×10^{-5}) with standard deviation of 8.79×10^{-4} (7×10^{-5}). The measurement is shown in Fig. 27. This pertains to a signal to noise ratio of 5:1.

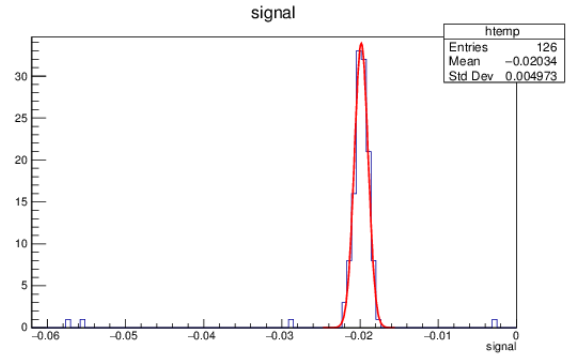


FIG. 27. The thermal equilibrium measurement used for the signal to background studies.

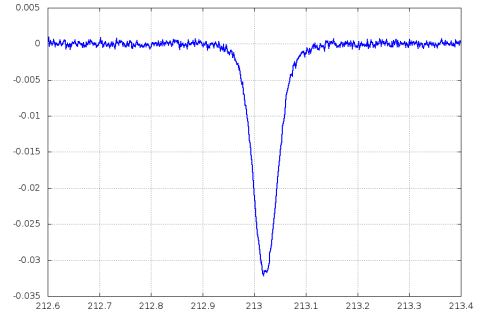


FIG. 28. The thermal equilibrium measurement used for the signal to background studies.

The absolute noise level measure without the crystal is 1.3×10^{-3} .

Similarly the TE signal is used to study the signal to noise ratio as seen in Fig. 28. Around 700 measurements are taken during a steady state TE. The mean of the signal was -3.20×10^{-2} (6×10^{-5}) with standard deviation of 1.21×10^{-3} (5×10^{-5}). This pertains to a signal to noise ratio of 6:1. Testing various coils windings and diameters results in range from 5-10:1. The absolute noise level measure without the TE is $1 - 3 \times 10^{-3}$. The larger coils or greater number of windings will naturally couple to more material, but they also are larger antennas and susceptible to noise.

An additional test was run with three coils in an 8 cm long target cell with TE studied at every coil. The coils were of similar dimension as previously described and spaced approximately 2 cm apart as shown in Fig. 29. The NMR was run at each coil individually and found no difference at this distance in running each coil independently with the other coils removed. In addition two coils were run simultaneously to find interference from the RF in both systems. The amplitude of the RF signal can be seen in Fig. 30. The dynamics of the interference is dependent on the sweep rate but this additional noise was able to be nearly mitigated when using the coils that were furthest apart (~ 5 cm) and taking a high statistics baseline (~ 5000 sweeps) with the opposing NMR sys-

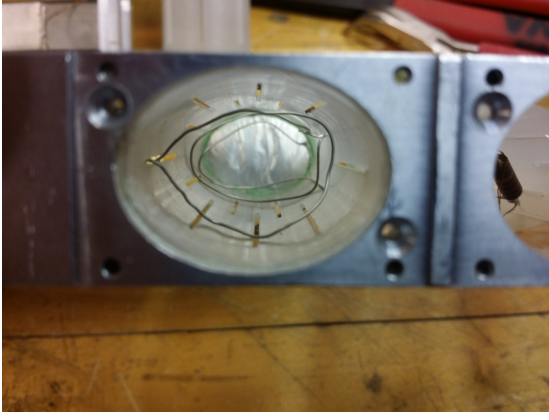


FIG. 29. Image looking down the mouth of the target cell with three NMR coils.

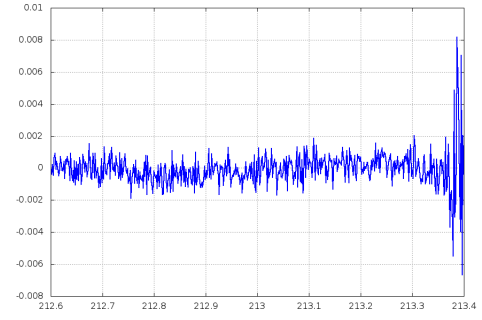


FIG. 30. The noise generated by the other RF-coil running simultaneously.

tem running. More work is required but this initial look may indicate the possibility of running two coils simultaneously for the dual helicity system. The longitudinal field for the Hall-B horizontal configuration makes the saddle coil the most desirable NMR coil geometry but maximizing inductance while minimizing cross talk will be an essential component of the setup if simultaneous measurement is chosen.

V. CONCLUSIONS ON PRELIMINARY STUDIES

COMSOL has proven itself to be an incredibly capable and robust software suite that truly gives the user the ability to prototype a model for most experiments entirely in the computer.

(This section should include conclusions on the physics not on COMSOL. We need to summarize what we learned from the simulations and tests and where things should go from here.)