newFASANT

Monostatic RCS Simulation with MONCROS

Benchmark: Monostatic RCS Simulation with MONCROS
Software Version: 6.2.7
Date: 24 Nov 16:26

Index

1. Benchmark Description and Objectives

This benchmark shows how to set up and analyze RCS simulations using the MoM approach. More specifically, different choices for the parameter values that can be found under the solver menu are analyzed and explained, showing the differences in simulation performance that are derived from the use of such values. Images and figures of the geometry and results are provided as well.

In this example we will start by considering a realistic geometry of a car located upright on the XY plane, including material definition for the glass and tires. The monostatic RCS will be obtained for an angular sweep around the car. The simulation parameters will then be fine-tuned in order to obtain good convergence behavior while maintaining low memory requirements and CPU time.

Default parameters are kept unless otherwise indicated.

This example is generated using the **MONCROS** module, so create a new project by clicking on the **New Project** button and selecting the MONCROS option.

1.1. Importing the geometry and setting up the layers

There are some important elements that should be taken into account for the geometric modelling:

- **Units:** It is recommended to work with the Units that fit the most to the dimensions of the geometry, as some parameters are automatically set-up. The following functions are especially sensitive to the working Units:
  - **Mouse functions:** such as zoom and selection functions. If any difficulty is found for these purposes, select the approximated elements on the desired area or from the Tree, click on Reset View button and the click on Zoom Selected button. Then, the camera functions are centered on the selection to improve the mouse functions near this region.
  - **Boolean operations:** such as split or projection commands. It is recommended that the Units are similar to the smaller elements involved in boolean operations, as the accuracy may be improved.
  - **Meshing process:** advanced meshing parameters such as the topology detection factors are predefined to the working Units.
- **Reference Plane:** This utility is useful to place and orientate correctly any geometric element. If the reference plane local coordinates are enabled, every new command will generate the new object according to the reference system.

In this example, we can consider that the default **Units** and **Reference Plane** are valid.

In this application, we are going to import an existing geometry. It may have been generated using an external tool, or it can be generated with the CAD modelling tools included in newFASANT. Such geometries can be defined with different formats, such as NUR files (newFASANT native format), IGES, STEP, DXF, MSH, STL, NAS, or others.

In order to import a geometry go to **Geometry – Import** and select the format and the file. Fig. 1 shows the model after being imported.

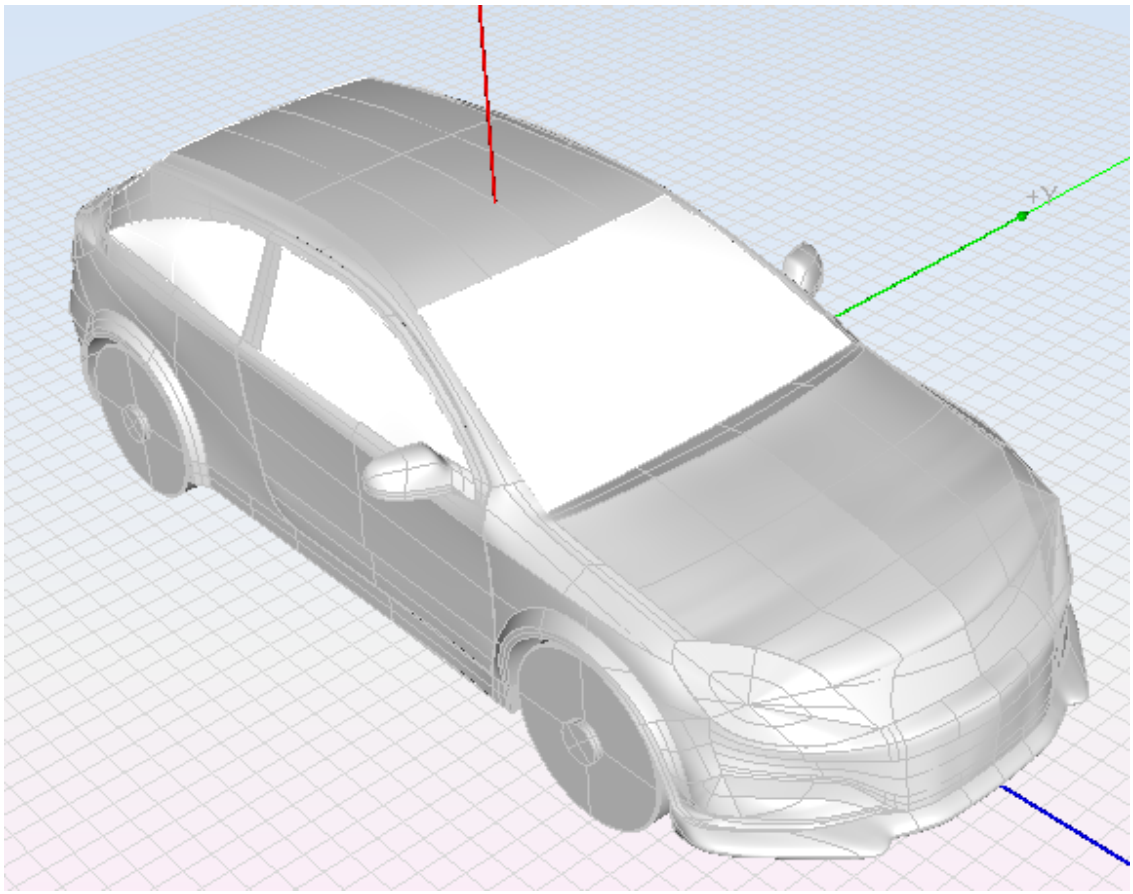

Figure 1: Imported Geometry

After this procedure, and in order to assign material properties to different surfaces of the model, it is recommended to organize the model in terms of layers. Each layer is a group of surfaces and can be assigned different colors for the visualization. Using the layer manager (**Edit – Layers – Manage Layers**) it is very easy to enable or disable the visualization of any layer, allowing to inspect parts of the geometry without the graphical clutter introduced by the surrounding surfaces. Note that using layers only affects the visualization of the geometry, but not the simulation itself. The distribution of the surfaces of the geometry in terms of layers for this example is shown in Fig. 2. Different layers have been assigned to the body of the car, the chassis, the glass surfaces (including windows and mirrors) and the wheels.

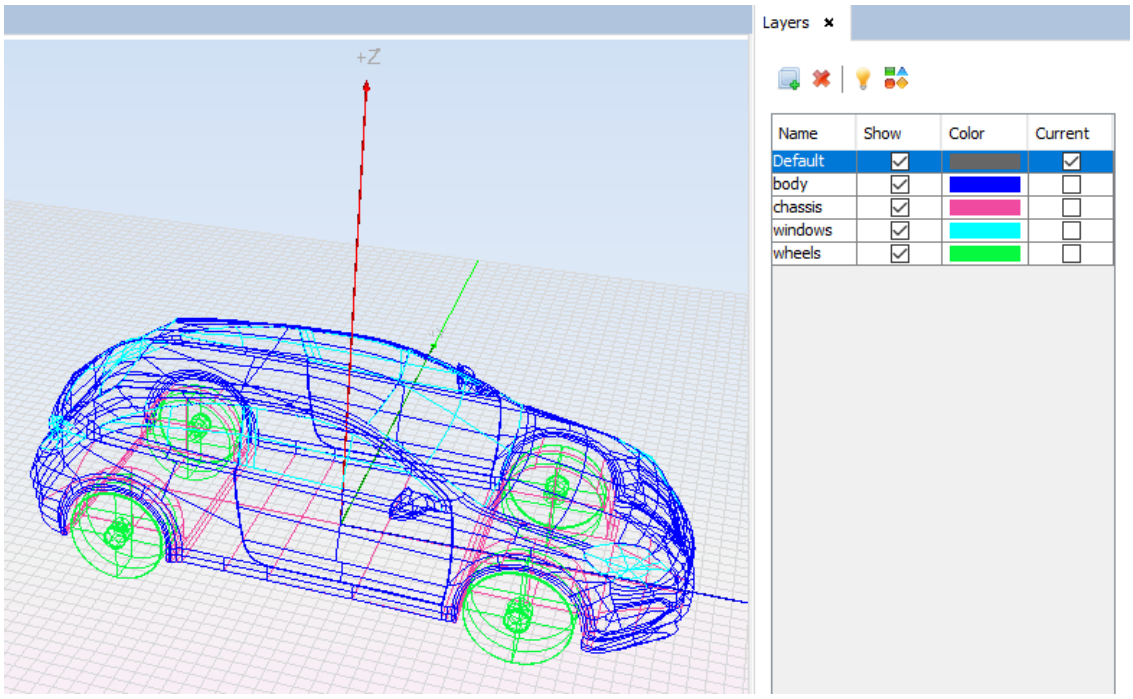

Figure 2: Classification of the surfaces of the model in terms of layers

1.2. Assigning material properties to the geometry

After setting up the layers the materials assignment to the surfaces of the model is very straightforward because it becomes very easy to select all the surfaces of a given layer without picking any of the other parts of the geometry. The first step in this section is the definition of the materials to be assigned, in case that they do not already exist in the material database. For this example, we are going to consider glass, with an electric permittivity $\varepsilon r = 4$, and rubber compound for the tires, with electric permittivity $\varepsilon r=7$ and $\tan(\delta)=0.04$. To do so, go to **Materials – Add** and introduce the name of the material under Material Attributes and the permittivity and/or losses values clicking on Set Parameters. Fig. 3 shows these values for the rubber compound previously mentioned. Note that it is possible to assign a certain color for the visualization of the material on the screen later. After introducing the parameters press **Save**.

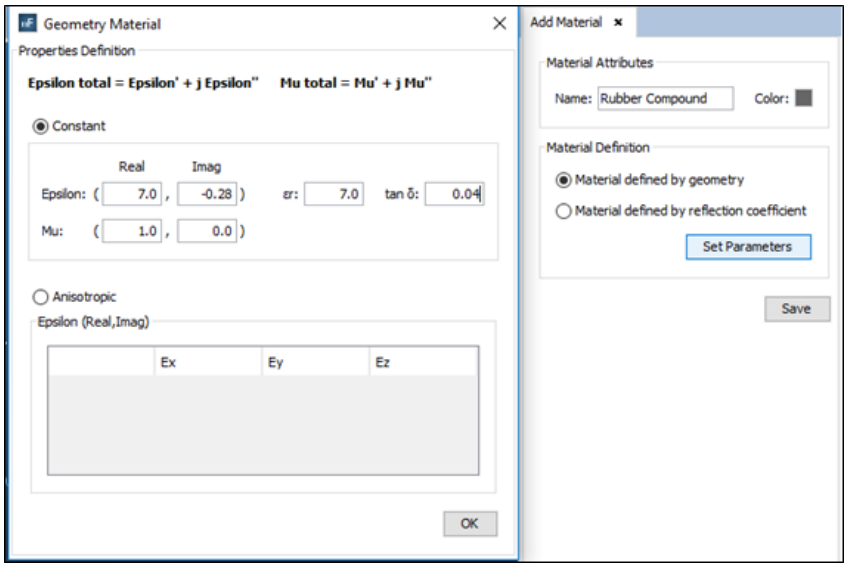

Figure 3: Adding a new material to the database

After including both materials, it is possible to assign them to the surfaces by selecting the layer manager and mark only the "Show" checkbox of the Wheels layer. After that select all the surfaces and click on Materials – Assign. In this example, we can select the material defined by geometry, and from the drop menu picks "Rubber Compound", configuring the thickness of the material as well. Note that for small fractions of the wavelength the thin layer approximation is adequate and a superficial mesh and simulation can be performed. Otherwise, a volumetric mesh and analysis may be necessary. Fig. 4 illustrates the procedure commented above about the material assignment.

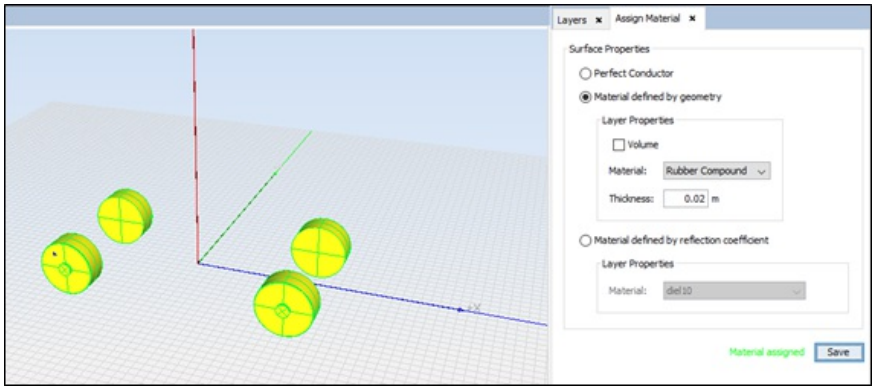

Figure 4: Assigning a new material to the surfaces contained in a layer, with a thickness of 2mm

2. Set-up description

The main parameters recommended for the solution of this benchmark case are presented in this section.

2.1. Simulation Parameters

The main simulation parameters are resumed in this section, such as the information about the units, materials and observation points/directions.

Regarding the Units, the default units (meters) are appropriate for the current example, as the length of the car is about 4.2 meters. Other value for the Units parameter would be equally valid for this example, but the axis and reference plane sizes should be scaled to the selected units. In the case of changing the current *Units*, remember to click on the YES button to the message that requires scaling the geometry.

Click on **Simulation – Parameters** to open the Simulation panel. The only parameter which has been modified for this example is the *initial frequency* that is set to **2.5 GHz**, which is the frequency that will be considered for the RCS calculation.

Click on **RCS – Parameters** to set up the excitation and the type of RCS to be computed. Leave the default options checked: Monostatic RCS, and the incident polarization following the Etheta component.

2.2. Solver Parameters

The recommended values for the solver configuration are explained in this section.

This example may be solved with the default parameters. The most important parameters within the **Solver – Parameters** panel are listed:

- **Solver Method:** The *Method of Moments (MoM)* is a full wave analysis method and will be considered for this benchmark. *Physical Optics (PO)* is faster and requires less memory, but should only be used in those cases involving large and smooth objects, due to its asymptotic nature.
- **Architecture Strategy:** Both *MPI* and *OpenMP* provide similar results whenever the machine supports them. OpenMP is not compatible with distributed memory systems, although it can be more efficient for shared memory systems such as workstations.
- **Electromagnetic Equation:** the only option that can be applied in the present case is EFIE because *MFIE* and *CFIE* require a closed conducting volume with its normal vectors pointing outwards.
- **Solver Functions - Advanced Options:** Use this window to modify any advanced configuration of the Solver. For analyzing problems similar to the considered one, the Preconditioner may be used to notably improve the convergence of the iterative process and reduce the simulation time.
- **Other Parameters:** The *Relative* error may be reduced to ensure a higher accuracy in the resolution of the problem. At most an error of 0.01 (or even smaller) is especially recommended when the *SAI preconditioner* is selected.

The configuration shown in the next figures has been used to analyze this benchmark.

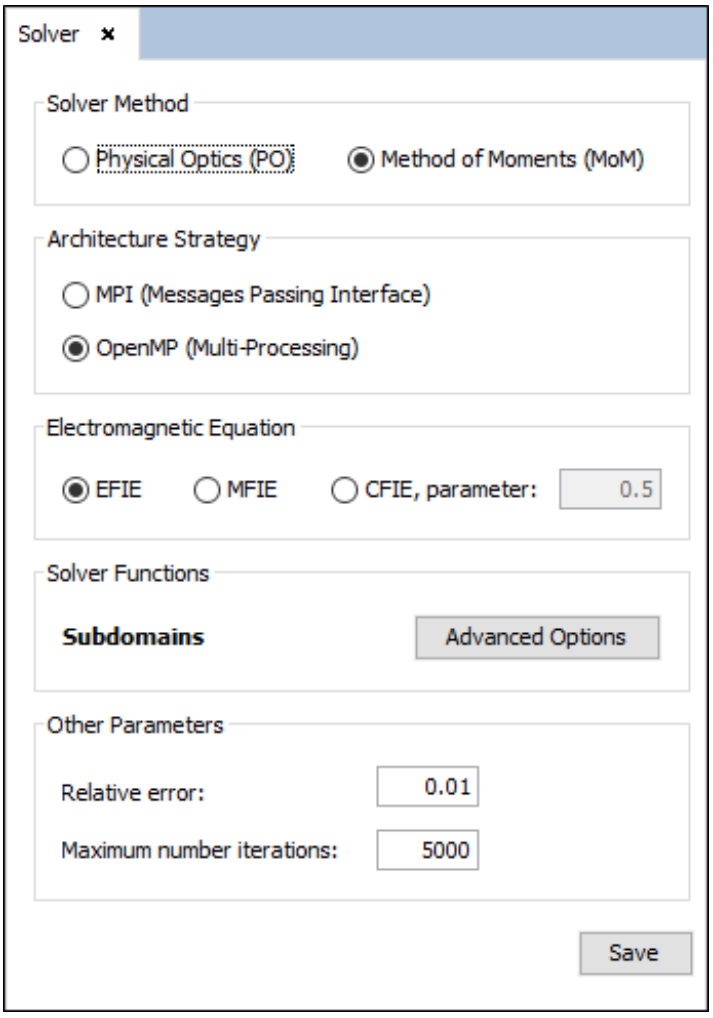

Figure 5: Solver Parameters – Main parameters

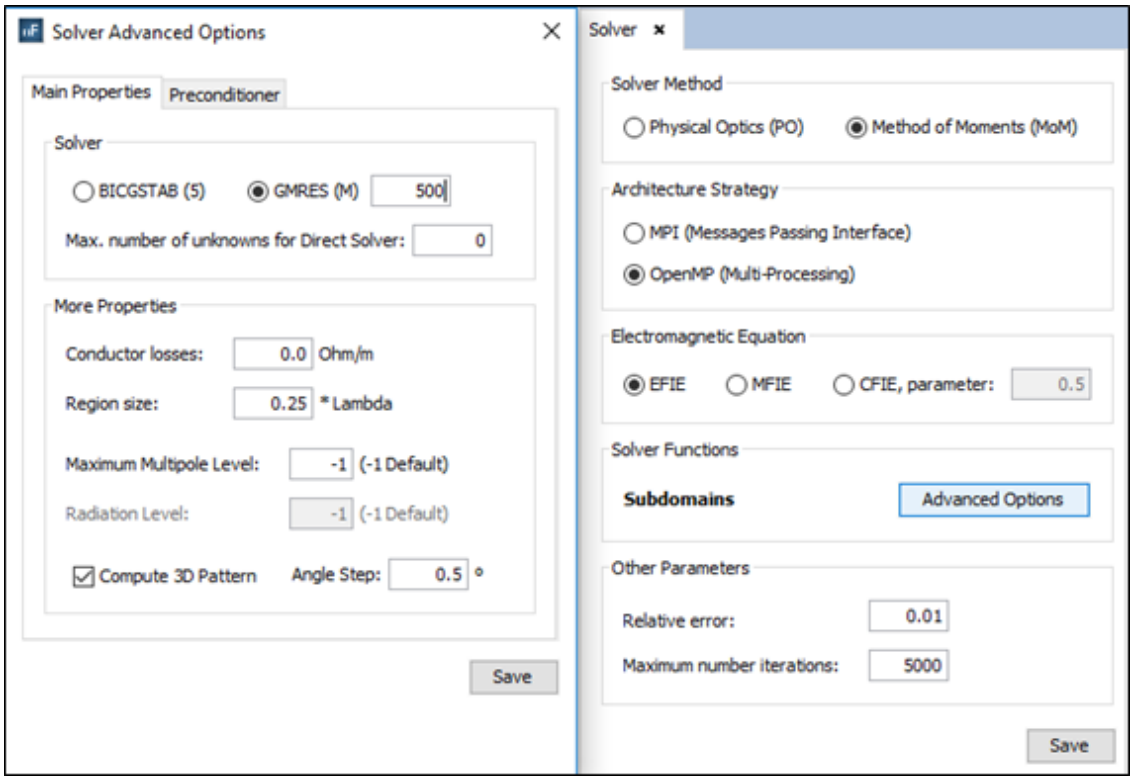

Figure 6: Solver Parameters – Advanced Parameters

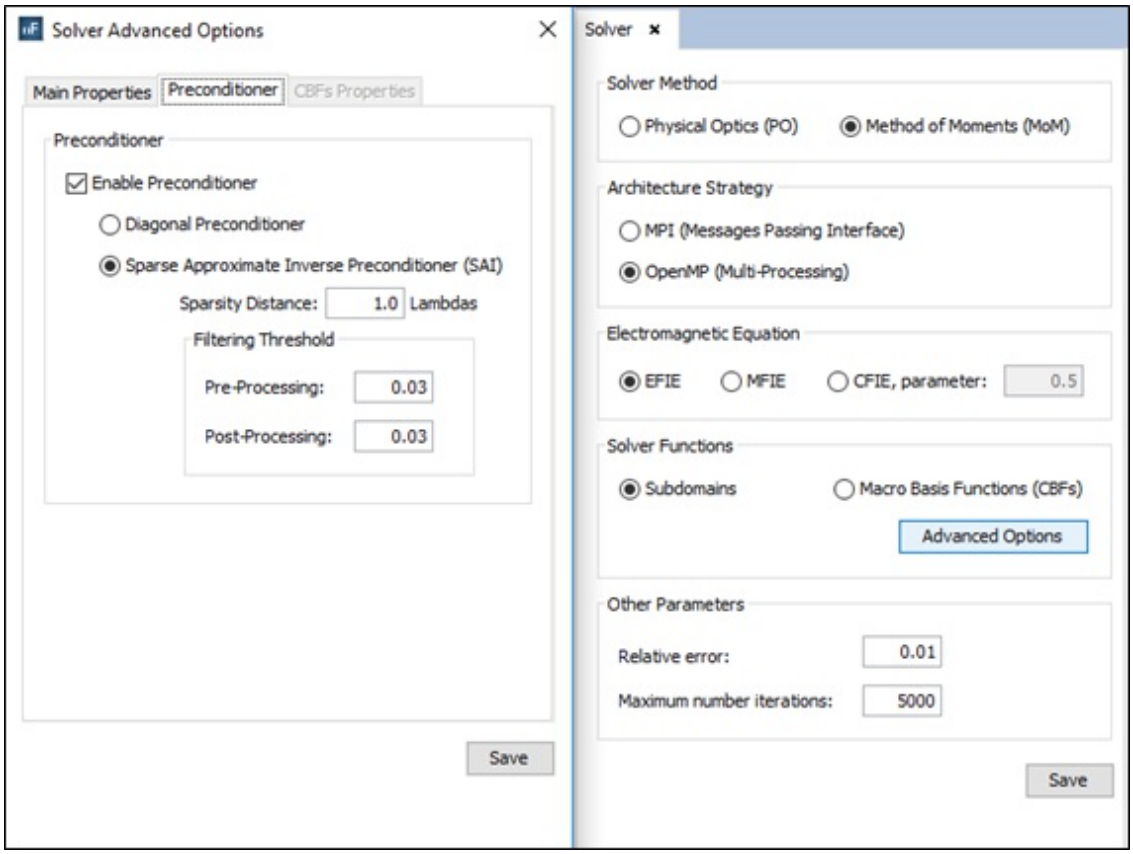

Figure 7: Solver Parameters - Preconditioner Parameters

2.3. Meshing Parameters

For this example the number of processors that has been set to 4, using 8 divisions per wavelength for planar and curved surfaces.

The most interesting parameters for this kind of problems are:

- ***Divisions per wavelength***. Increase them for obtaining denser meshes or reduce them for having fewer unknowns. 10 divisions are large enough for problems such as the considered one, and less than 5 divisions are not recommended as it may be detrimental to the Method of Moment's formulation accuracy.
- ***Mesh Mode.*** Modify it for frequency sweeps, and then you can use a mesh generated for the same frequency for all the analysis frequencies, or generating multiple meshes that are assigned to the closest analysis frequency.
- ***Advanced Settings – Multilevel Meshing Mode.*** Disable this option when the expected mesh density is not very large for the input surfaces, i.e., for meshing problems electrically small. For large cases where the input geometry is analogous to a mesh, such as periodical structures or FSS elements, the Multilevel Mode may be also disabled. It is recommended to use the multilevel mode when a large number of elements is expected in any of the input surfaces. You can also specify the frequency for generating the mesh in the first level, and this mesh will be remeshed up to the final desired frequency.

Use the command ***area –all*** to get information about the total area to be meshed. For this example, it is about 28.49 square meters, so we can estimate that about 126,521 elements must be generated (squares with a size of the wavelength divided by the specified divisions) to mesh the geometry. The estimated number of unknowns to be solved is about 253,042 (approximately the double of the mesh elements). According to the estimated resources, this example requires the ***Gold Version*** to be solved.

This example has been meshed in a personal computer by using 4 processors and requiring approximately 900 MB of RAM and 3 minutes to obtain a mesh of about 137,000 elements.

2.4. Configuration of the Output Results

Use the ***Output – Observation directions*** menu to add a cut with phi ranging from 0 to 180 degrees for theta = 0 degrees, as shown in Fig. 8. The default angular sweep can be deleted.

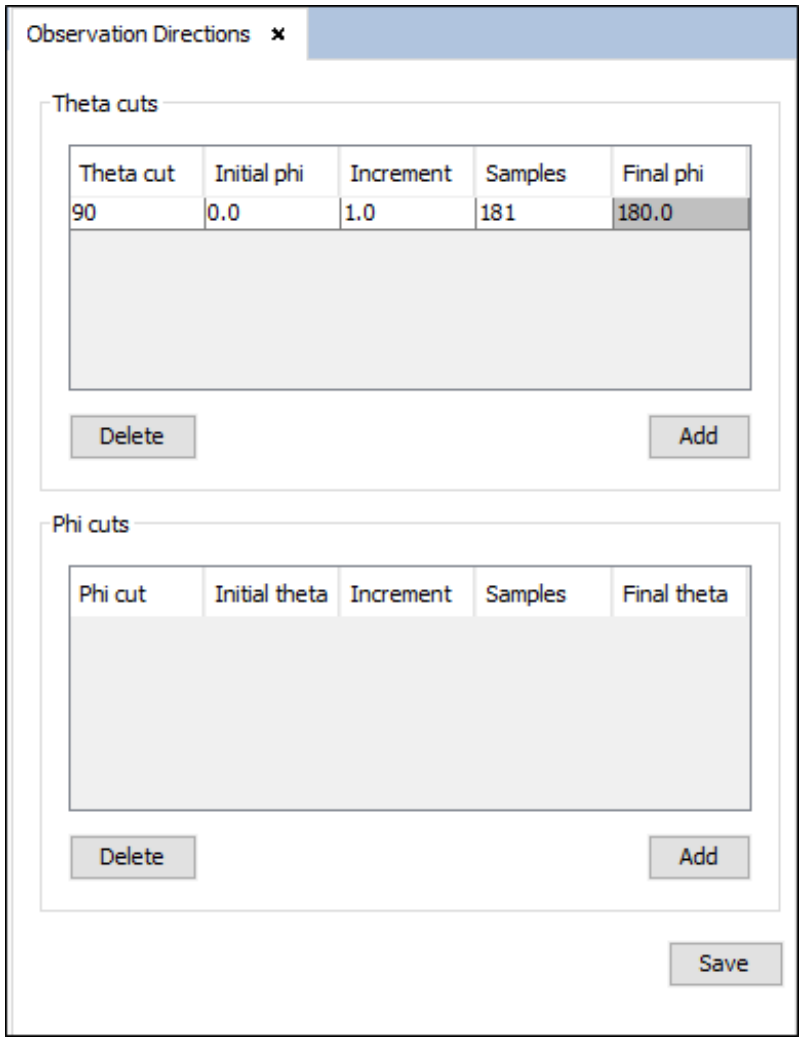

Figure 8: Configuration of the observation directions

3. Visualizing the Results

At this point, we can launch the simulation. Monostatic analyses require much more time than bistatic cases because the system must be solved for each observation direction. In order to speed up the solution time, we launch the simulation using a workstation with 16 computing cores. A conventional computer is perfectly able to obtain the same results, although requiring longer processing time.

The computed monostatic scattered field for this problem required about 13h and 40m, with an average of 104 iterations to solve each incidence direction using the *GMRES* solver. The resulting RCS plot is shown in Fig. 9, and can be visualized selecting the **Show Results – Far Field – View Cuts** option.

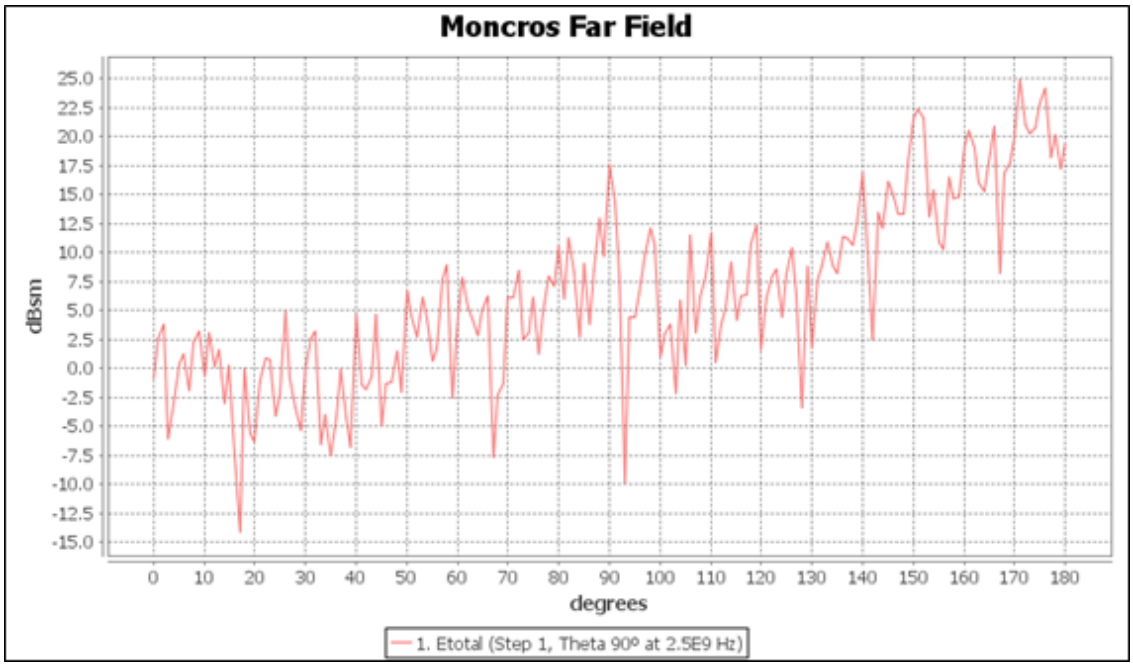

Figure 9: Monostatic RCS results for the angular sweep

Note that for monostatic computations the current visualization plots are not available since for each incident direction there is a different current distribution. In order to visualize the current distribution, a bistatic analysis must be performed.

4. Computational Resources

The benchmark includes a resume of the computational resources required in order to obtain the results shown above.

CPU type		Workstation Memory + Processors	
Resources	**Number of processors**	**RAM required (GB)**	**Time Required (mm : ss)**
HP Z820 Workstation	16	5	820 : 23