Ansys
2025 R1
POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with HFSS™: RCS Test Model (Ogive)

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Conventions Used in this Guide

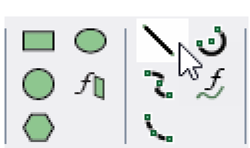
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, "click **HFSS** > **Excitations** > **Assign** > **Wave Port**."
  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *filename*" means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press Shift+F1" means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"

This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

Table of Contents **Contents-1**
1 - Introduction **1-1**
RCS Test Model (Ogive) 1-2
Set Up the Project 1-2
Enable Legacy View Orientations 1-3
2 - Create the Ogive **2-1**
3 - Add an Incident Plane Wave **3-1**
4 - Assign an IE Region **4-1**
5 - Add the Solution Setup **5-1**
6 - Validate and Run the Simulation **6-1**
7 - Plot and Animate the Current **7-1**
8 - Plot the RCS **8-1**
9 - Optionally, Restore Current View Orientations **9-1**

1 - Introduction

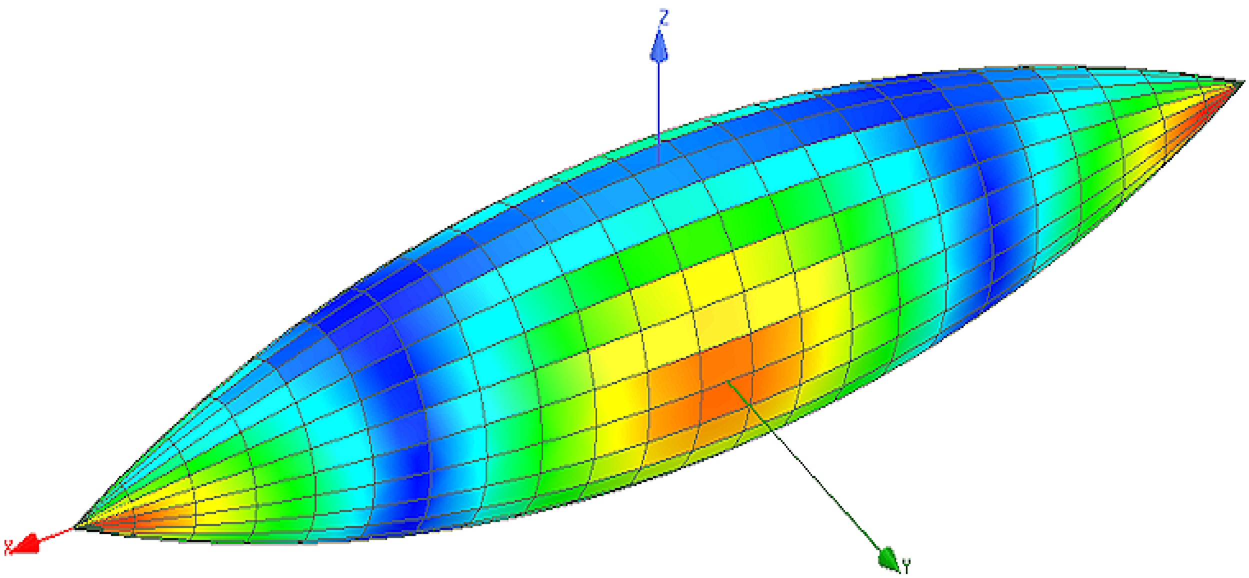
This example looks at the scattering from a standard Radar Cross Section (RCS) test model (specifically, the ogive) using the HFSS design type with an IE region assigned to the ogive. An IE region results in a full-wave Integral Equation solver being used to analyze the model. This solver is particularly well suited for analyzing large conducting objects, and it eliminates the necessity of creating an open region (that is, a vacuum or air box) around the model to determine far field effects.

> **Note:**
>
> IE Regions in an HFSS design have replaced the legacy HFSS-IE design type. If you open a legacy HFSS-IE design, you are prompted with the option to automatically convert the model to an HFSS design with an IE Region applied to its objects.

You will perform the following tasks to complete this exercise:

- Create a new project, insert an HFSS design, and set the solution type
- Enable Legacy View Orientations
- Create the ogive by drawing a segmented equation based curve and sweeping it in specified increments around the X-axis
- Define an incident plane wave
- Assign an IE region
- Add the solution setup
- Run the simulation
- Overlay and animate the surface current magnitudes
- Plot the monostatic RCS results

RCS Test Model (Ogive)



We will look at the scattering from a conducting ogive with a length of 10" and a half angle of 22.6° (0.3948 radians). The ogive lies with its axis along the global X axis. The equation that defines this model is:

$$F(x) = \sqrt{1 - [x/5 * \sin(22.6°)]^2} - \cos(22.6°)$$

$$y = F(x) * \cos(\varphi) / [1 - \cos(22.6°)]$$

$$z = F(x) * \sin(\varphi) / [1 - \cos(22.6°)]$$

For this exercise, x, y, and z are defined in inches.

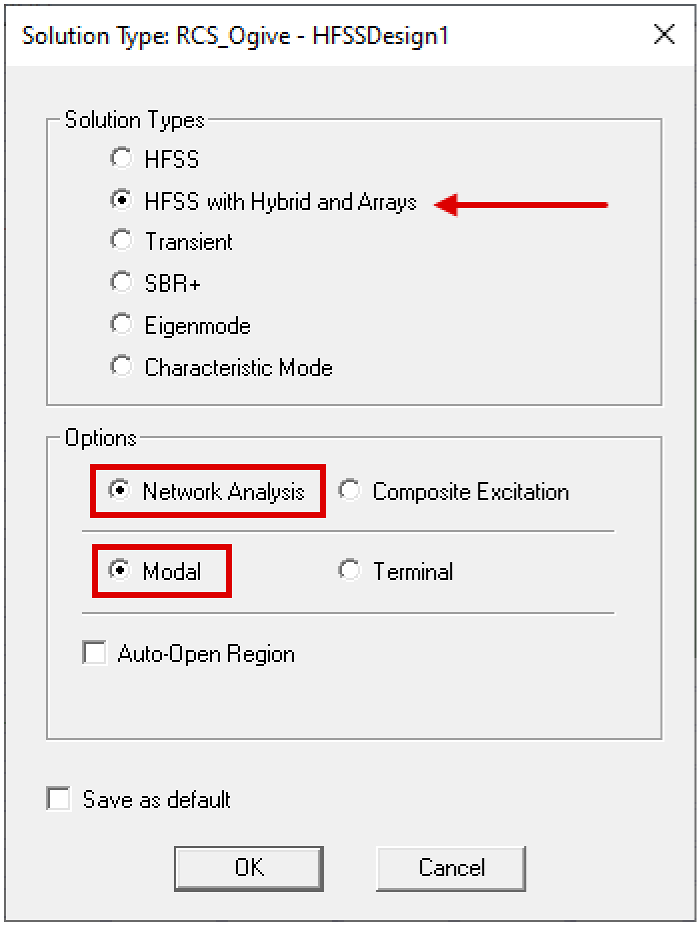
Set Up the Project

Creating this model is very straightforward. We will create an equation based curve in the XY plane and then sweep the curve around the X axis to create the 3D solid. The curve will be segmented (that is, comprised of a series of short straight segments). Similarly, it will be rotated about the X-axis in discrete increments, producing a discrete approximation of the ogive shape.

1. Launch the Ansys Electronics Desktop application.

   A new project is automatically created when you launch the application. If AEDT was already running with no project open, click **File** > **New** in the menu bar.

2. Insert an HFSS design into the new project (**Project** > **Insert HFSS Design**).
3. From the menu bar, click **HFSS** > **Solution Type**.
   a. In the dialog box that appears, choose the solution type and options shown in the following image:



b. Click **OK**.

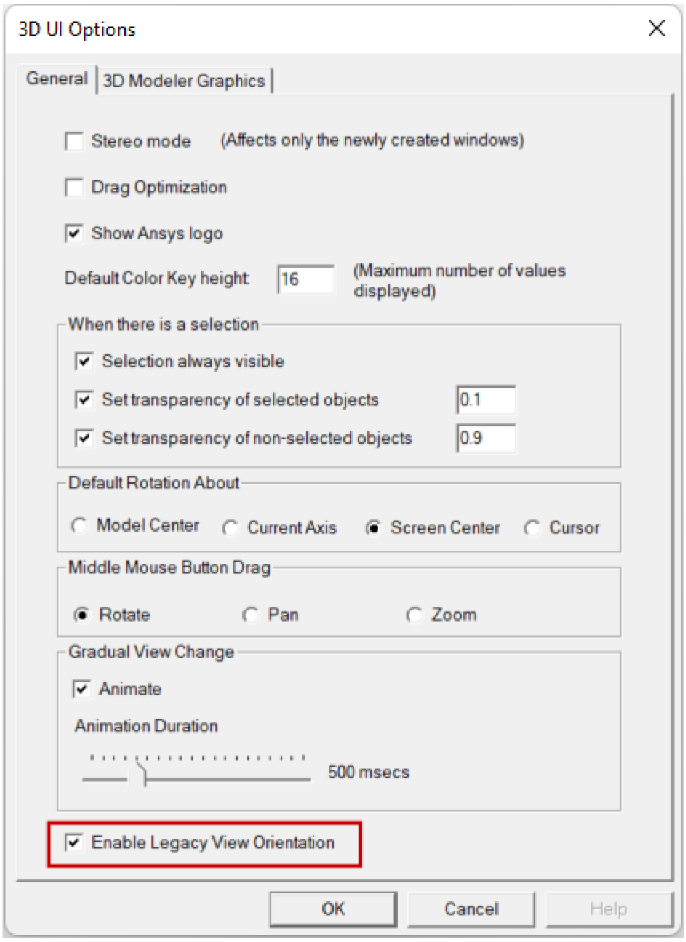
Enable Legacy View Orientations

This getting started guide was created based on standard view orientations that were in effect for version 2023 R2 and earlier of the Ansys Electronics Desktop application. For consistency between your experience and the views and instructions contained in this guide, select the *Enable Legacy View Orientation* option in the 3D UI Options dialog box, as follows:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Select **Enable Legacy View Orientation**:



3. Click **OK**.

   Changing the view orientation option does not change the model viewpoint that was in effect at the time.

4. On the **Draw** ribbon tab, click **Orient** to change to the *Trimetric* view, which is the default legacy view orientation.

   You do not have to select *Trimetric* from the *Orient* drop-down menu. The default view appears when you click *Orient*.

Although this option can only be accessed once a design is added to a project, it is a global option. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

At the end of this guide, you will be prompted to clear the *Enable Legacy View Orientation* option, if you prefer to use the view orientation scheme implemented for 2024 R1 and newer versions going forward.

For a comparison of the legacy and current view orientations, search for *"View Options: 3D UI Options"* in the HFSS help. Additionally, views associated with **Alt + double-click** zones have been redefined. The current orientations are shown in the help topic, *"Changing the Model View with Alt+Double-Click Areas."*

2 - Create the Ogive

Creating this model is very straightforward. We will create an equation based curve in the XY plane and then sweep the curve around the X axis to create the 3D solid. The curve will be segmented (that is, comprised of a series of short straight segments). Similarly, it will be rotated about the X-axis in discrete increments, producing a discrete approximation of the ogive shape.

1. Optionally, you can change the default units to inches (in), but we will ensure the proper units when defining the model. So, you can leave your units at the current default setting.

   If you want to change the default units, do the following:

   - On the **Draw** ribbon tab, click **Units**, select **in** from the **Select units** drop-down menu, and click **OK**.

2. From the **Desktop** ribbon tab, click **Save As**, navigate to the working folder of your choice, specify **RCS_Ogive.aedt** as the **File name**, and click **Save**.
3. On the **Draw** ribbon tab, click **Draw equation based curve**.

   The *Equation Based Curve* dialog box appears.

   Let X = the parametric variable (_t). In the XY plane, Z = 0, so the equation for Y is:

$$Y = \frac{\left\{ \sqrt{\left[\frac{1 - \_t * \sin(0.3948)}{5} \right]^2} \right\} - \cos(0.3948)}{[1 - \cos(0.3948)]}$$

   The angles are entered in radians and, to insure the units are handled properly, multiply each nonzero term (**X** and **Y**) by the quantity, **1in**. Therefore:

4. In the *Equation Based Curve* dialog box, you enter the equations as follows:

   X(_t): **_t*1in**

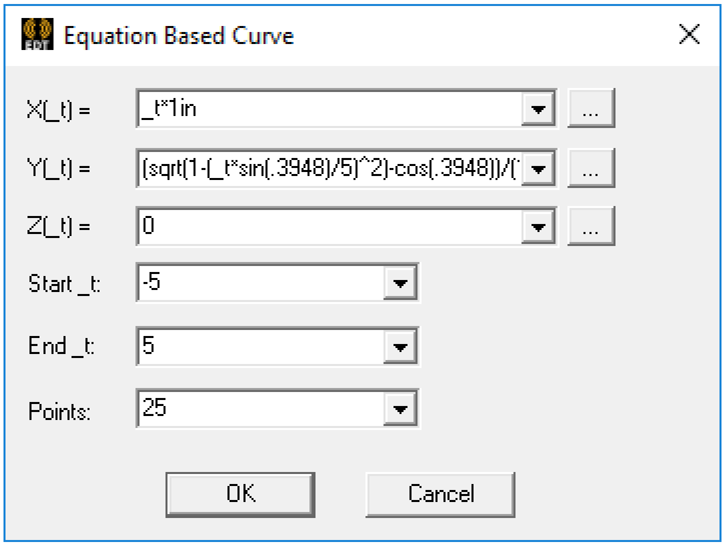
   Y(_t): **(sqrt(1-(_t*sin(.3948)/5)^2)-cos(.3948))/(1-cos(.3948))*1in**

   Z(_t): **0**

   Start _t: **-5**

   End _t: **5**

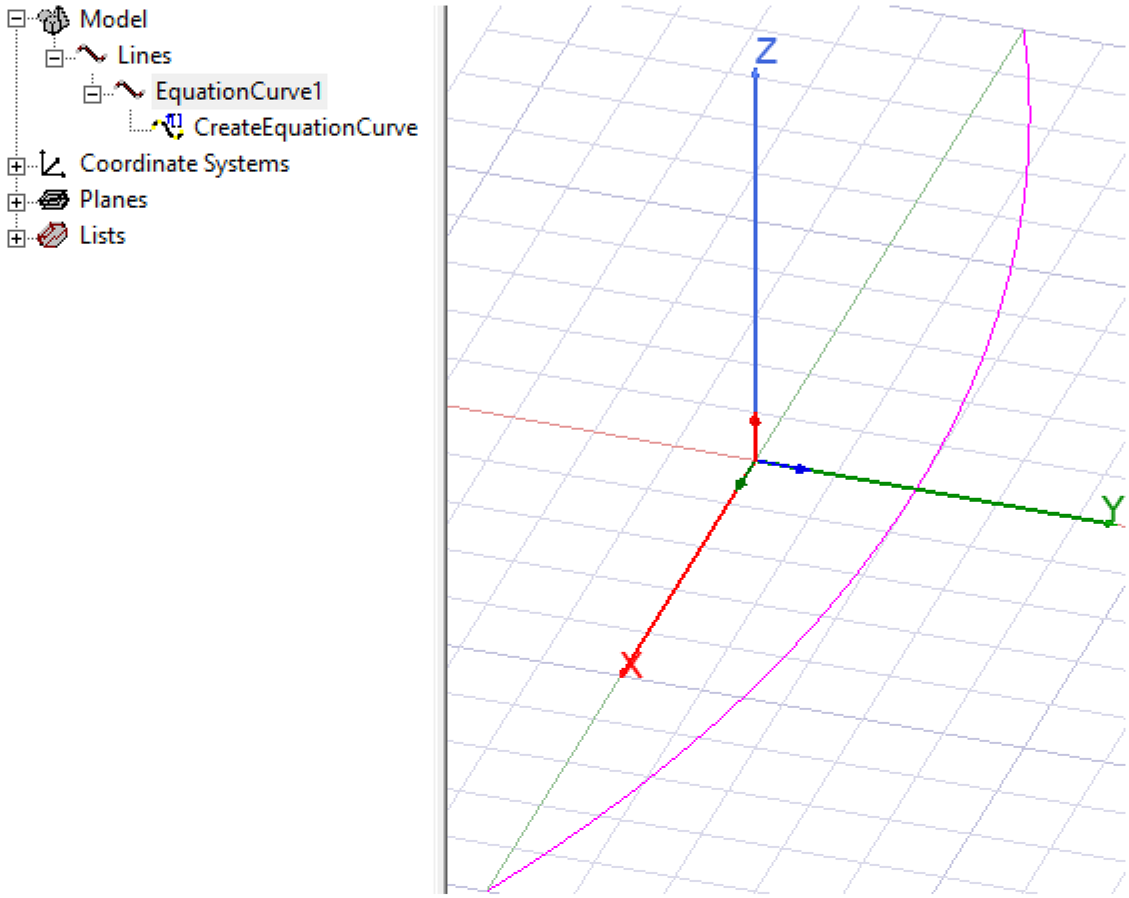
   Number of Points: **25**



> **Tip:**
>
> If you are not viewing a hard copy of this guide, you can copy the *X(_t)* and *Y(_t)* equations and paste them into the *Equation Based Curve* dialog box.

5. Click **OK** to close the dialog box and execute the command.

   You should see a curve like shown in the following image:



6. Under *Model* > *Lines* > *Equation Curve1* in the History Tree, double-click **CreateEquationCurve**.

   The *Properties* dialog box appears, allowing you to view the command settings with the *Value* column expanded for compete viewing, as shown below:

Name	Value	Unit	Evaluated Value
Command	CreateEquationCurve		
Coordinate System	Global		
X(_t)	_t*1in		*****
Y(_t)	(sqrt(1-(_t*sin(.3948)/5)^2)-cos(.3948))/(1-cos(.3948))*1in		*****
Z(_t)	0		0
Start _t	-5		-5
End _t	5		5
Number of Points	25		25

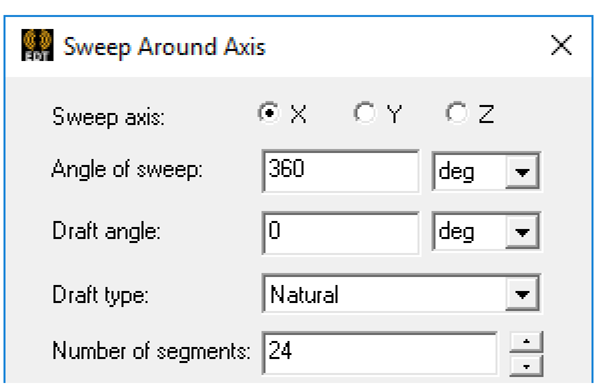
   If the curve does not display as expected, you can edit the *Value* cells here.

> **Note:**
>
> HFSS supports curvilinear elements (for smooth, unsegmented curves and surfaces) without restriction. However, you can also approximate curves with a series of discrete straight segments if desired. There are a couple of advantages to drawing segmented curves and surfaces:
>
> - A segmented surface helps the mesh engine to generate more consistent and uniform surface elements.
> - When you extrude a planar object (that is, sweep it along a vector or path) with a twist angle specified, the twist is much easier to visualize if the original object is segmented. Sweeping a segmented circular object with a twist produces helical line segments along the surface, whereas the result from a smooth circle would look just like a cylinder. An example of when you would use this type of geometry is an analysis of a twisted cable assembly.
>
> For this design, you will use a curve with *Number of Points* = 25 points (resulting in 24 segments). You will then sweep this segmented curve around the X-axis in 24 discreet segments to generate the full surface of the ogive.

7. Click **OK** to close the *Properties* dialog box.
8. In the History Tree, select **EquationCurve1**.
9. On the **Draw** ribbon tab, click **Sweep around axis**.

   The *Sweep Around Axis* dialog box displays.

10. Specify the following settings:
    a. *Sweep axis*: **X**
    b. *Angle of sweep*: **360 deg**
    c. *Draft type*: **Natural**
    d. *Number of segments*: **24**



11. Click **OK** to create the ogive and keep it selected.

    The resulting shape is a sheet object, since an open 2D curve was swept to make the ogive surface.

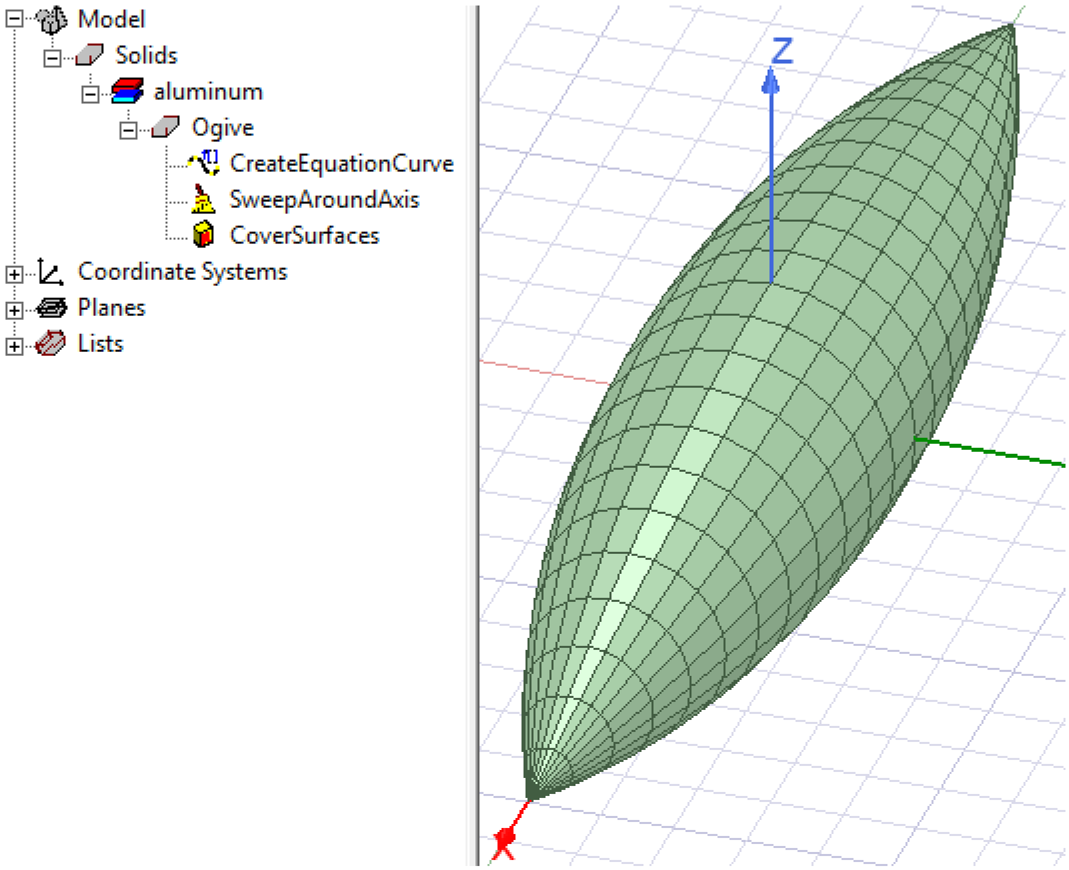
12. On the **Draw** ribbon tab, click **Surface** > **Cover Faces**.

    The ogive is converted to a solid object and listed under *Model* > *vacuum* in the History Tree.

    > **Note:**
    >
    > *Vacuum* is the default material for solid objects in an HFSS design. Since these objects are generally machined from aluminum, you will change the material in the next step.

13. While the ogive object (*EquationCurve1*) is still selected, make the following changes in the **Attribute** tab of the docked *Properties* window:
    a. Change the **Name** to **Ogive** and press **Enter**.
    b. Choose **Edit** from the **Material** drop-down menu, select **aluminum** from the listed library materials in the *Select Definition* dialog box, and click **OK**.
14. Press **Ctrl+D** to fit the model to the view area.
15. Click in the Modeler window's background area to clear the selection.

    Model and History Tree should now match the following image:



16. **Save** your project. (This command is available from any ribbon tab or from the **File** menu.)

The model geometry is finished. In a later step, you will assign an IE hybrid region to the ogive object. When analyzing an IE region, no open region (that is, vacuum or air box) is needed around the model to produce field or RCS results. Fields are computed on the surfaces of the conductors, and RCS results are computed without an open region using the full-wave Integral Equation solver.

3 - Add an Incident Plane Wave

You next need to add the radar source, in this case, an incident plane wave. You will compute the monostatic RCS in the XY plane. Since it is monostatic, you need to include many incident angles. For the scattering computed here, you should get a clean response if you solve using 3° steps.

1. With nothing selected, right-click in the Modeler window and select **Assign Excitation** > **Incident Wave** > **Plane Wave** from the short cut menu.

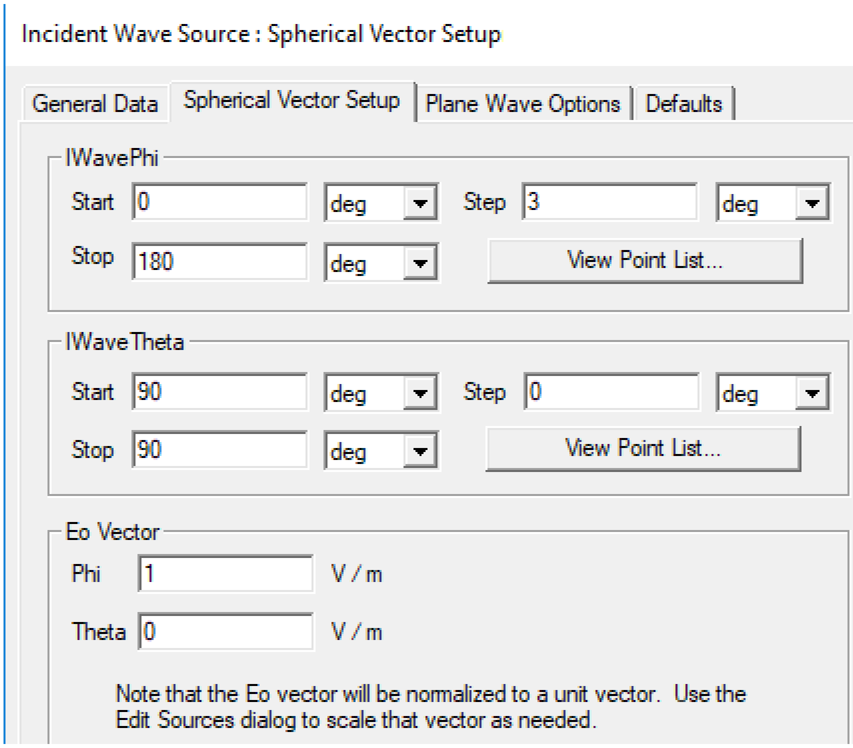
   The *General Data* step of the *Incident Wave Source* wizard appears.

1. Select the **Spherical** option for the **Vector Input Format** and leave the *Excitation Location and/or Zero Phase Position* coordinates at **(0,0,0)**.
2. Click **Next**.

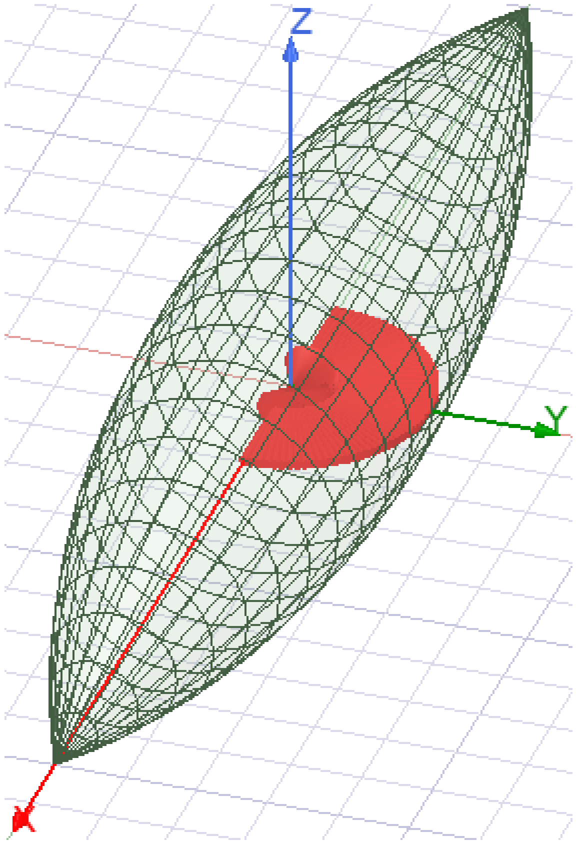
   The wizard advances to the *Spherical Vector Setup* step.

3. In the **IWavePhi** section, specify the following settings:
   a. *Start*: **0 deg**
   b. *Stop*: **180 deg**
   c. *Step*: **3 deg**
4. In the **IWaveTheta** section, specify the following settings:
   a. *Start*: **90 deg**
   b. *Stop*: **90 deg**
   c. *Step*: **0 deg**
5. In the **Eo Vector** section, ensure that *Phi* = **1 V/m** and *Theta* = **0 V/m**.

   The dialog box should match the following image:



6. Click **Next** and then **Finish**. The excitation is defined.
7. Under *Excitations* in the Project Manager, select IncPWave1 to visualize the incident plane wave you just defined:



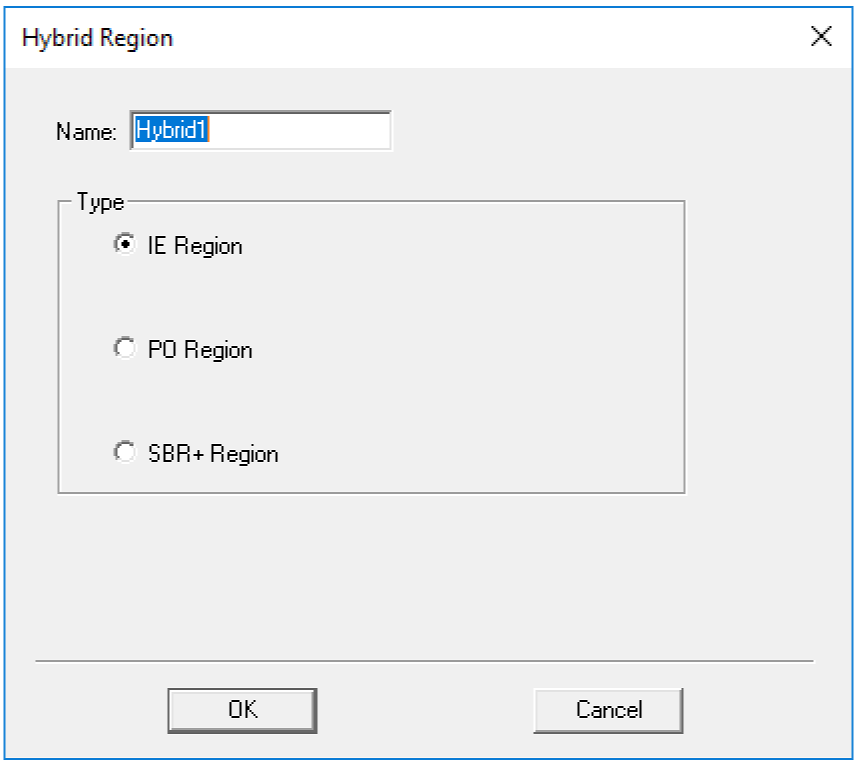
This setup will solve for the response due to incident planes waves at θ =90° and for 61 values between φ = 0° and φ = 180°, inclusively. The polarization is φ-directed, so we will be solving for the HH monostatic RCS.

4 - Assign an IE Region

Next, assign a hybrid region of the type *IE Region* to the ogive object.

1. Under *Model* > *Solids* > *aluminum* in the History Tree, select **Ogive**.
2. Right-click **Hybrid Regions** in the Project Manager and choose **Assign** > **IE Region** from the shortcut menu.

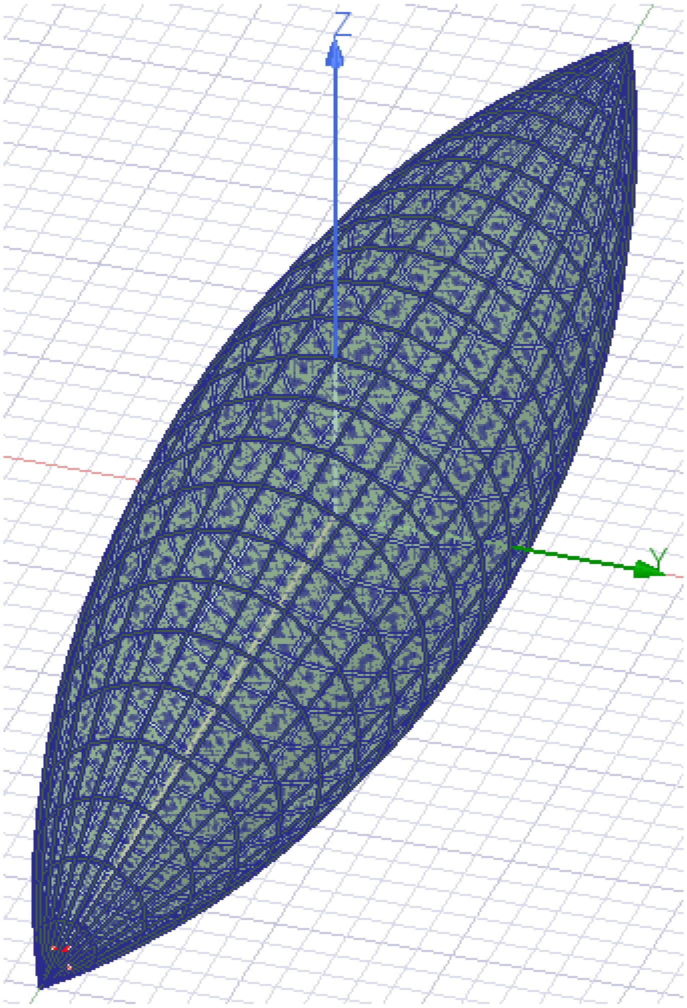
   The *Hybrid Region* dialog box appears:



3. Click **OK** to create the IE Region using the default type and name.

   Hybrid1 appears under *Hybrid Regions* in the Project Manager:

   Also, the IE Region is visualized on the model when selected:



4. Clear the selection.

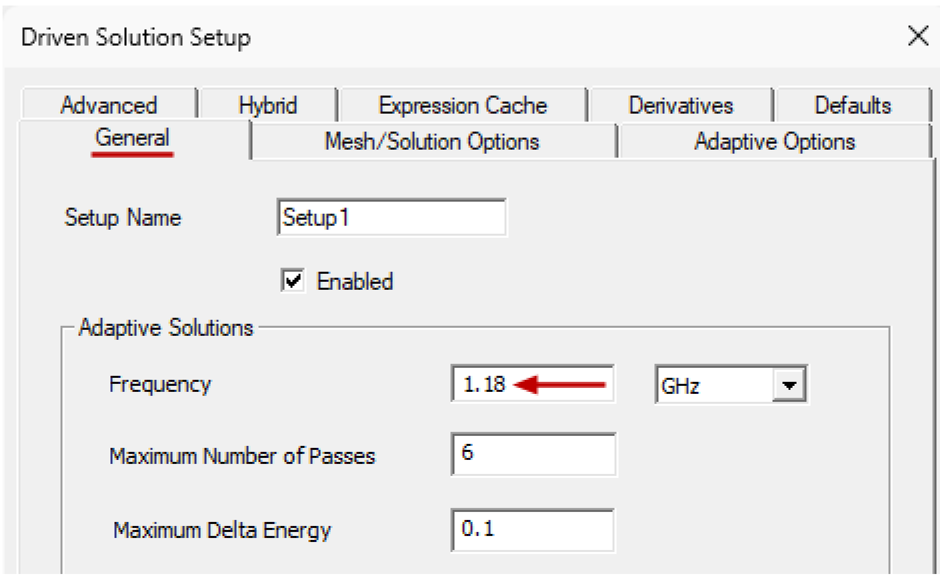
5 - Add the Solution Setup

The last step is to add the solution setup.

1. On the **Simulation** ribbon tab, click  **Setup** > **Advanced**.

   The *Driven Solution Setup* dialog box appears.

2. In the **General** tab, change the **Frequency** to **1.18 GHz**.



3. Click **OK**.

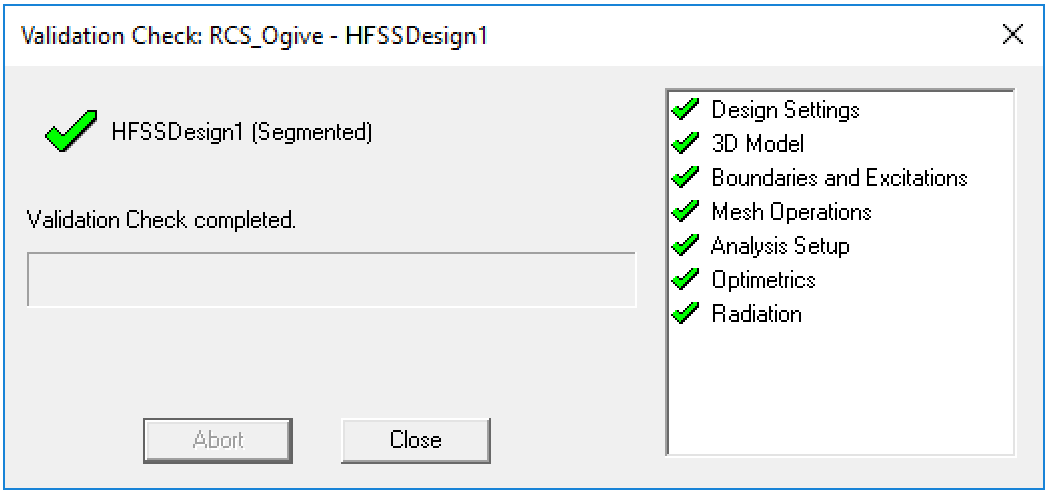
The Ogive solution setup is compete.

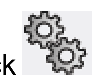
6 - Validate and Run the Simulation

Run a validation check to ensure that there are no errors and then run the simulation, as follows:

1. On the **Simulation** ribbon tab, click  **Validate**.

   The *Validation Check* window appears, and there should be no errors or warnings:



2. Click **Close**.
3. On the **Simulation** ribbon tab, click  **Analyze All**.

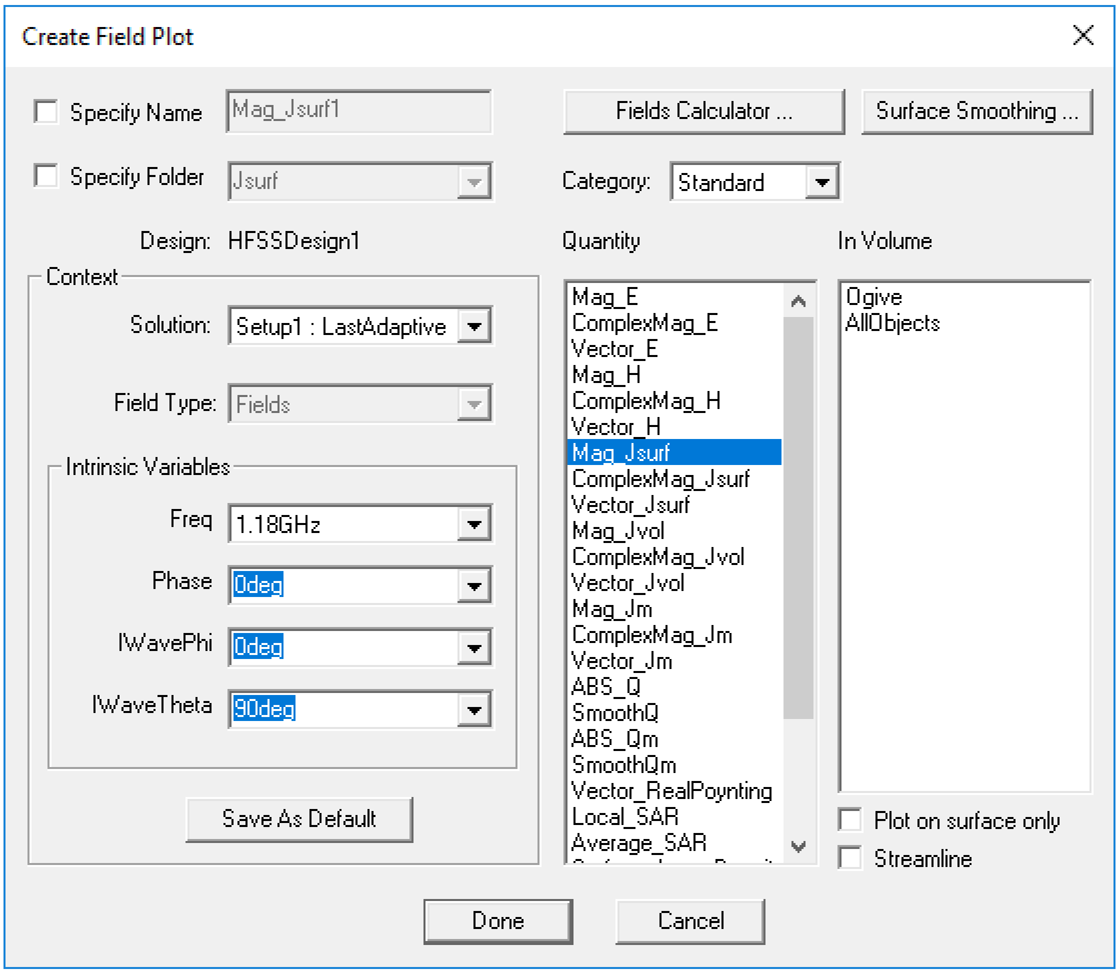
   The solution should converge in two passes and take a minute or less to run on a fairly current computer workstation.

7 - Plot and Animate the Current

To view the current pattern on the surface of the ogive:

1. Under *Model* > *Solids* > *aluminum* in the History Tree, select **Ogive**.
2. With the ogive selected, right-click **Field Overlays** in the Project Manager and choose **Plot Fields** > **J** > **Mag_Jsurf**.

   The *Create Field Plot* dialog box appears:



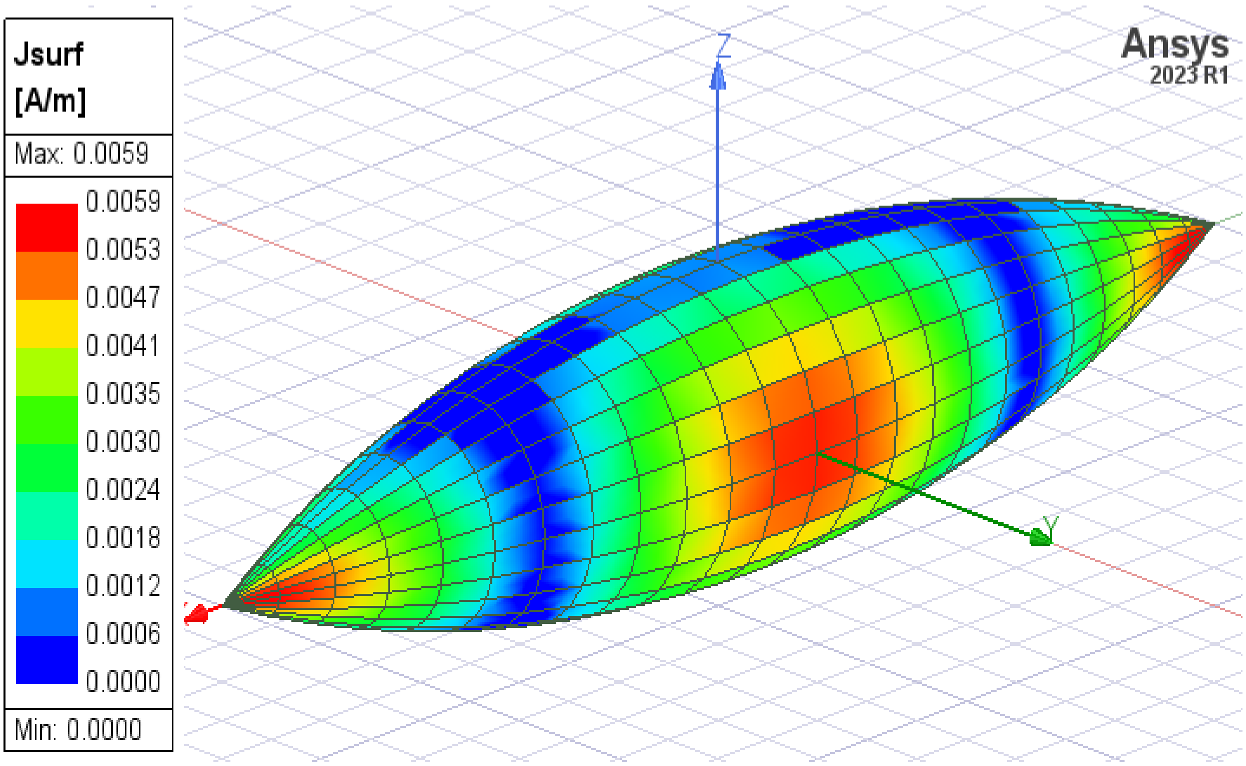
3. Click **Done** to create the plot using the default settings.

   You will see the induced current for the incident angle, φ=0° and θ = 90°.

4. On the **Model** ribbon tab, click **Orient** > **Dimetric** and press **Ctrl+D** for a better viewpoint of the current results.
5. Double-click within the perimeter of the plot legend to access the plot settings.

6. On the **Plots** tab, select **Gourard** from the **IsoValType** drop-down menu and then click **Close**.

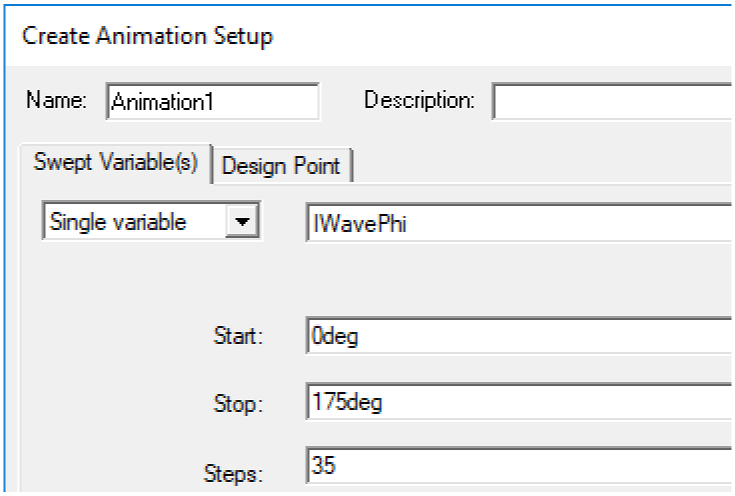
   This setting produces a smoother color shading effect, with gradual transition from one color to the next:



7. Under *Field Overlays* > *Jsurf* in the Project Manager, right-click **Mag_Jsurf1** and choose **Animate**.

   The *Create Animation Setup* dialog box appears.

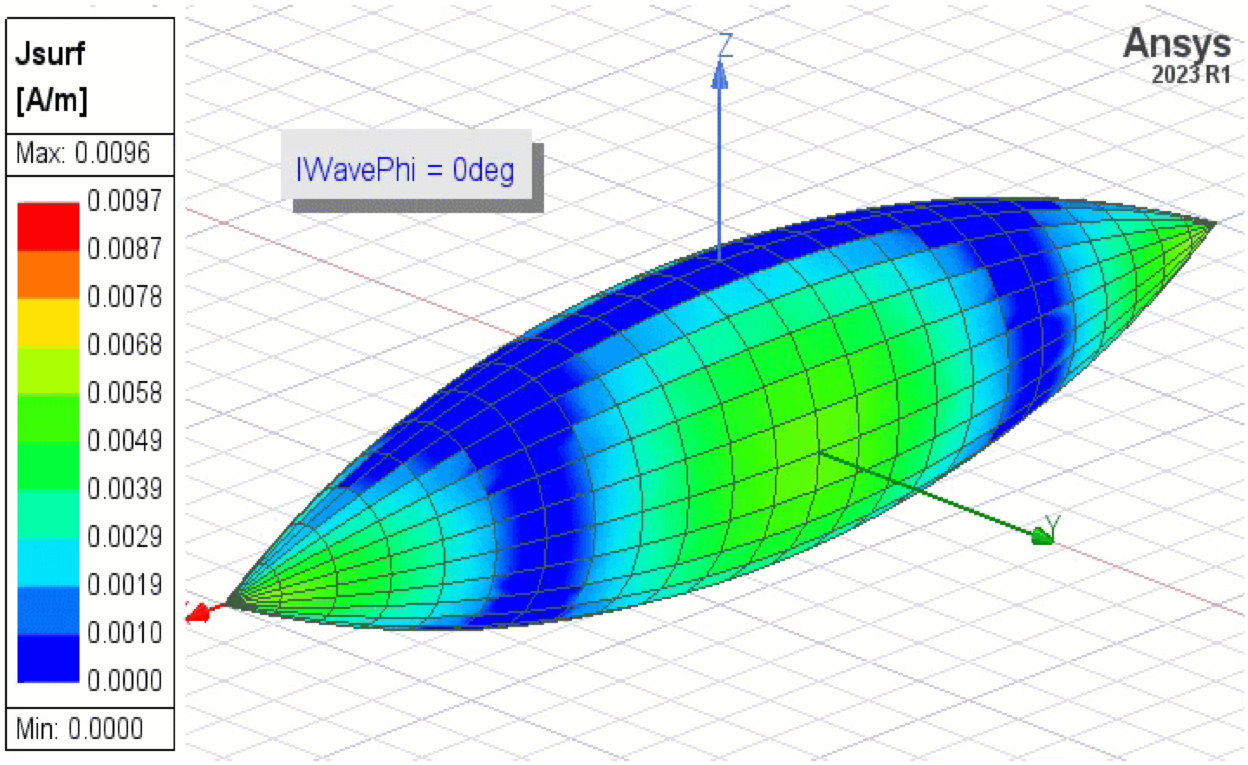
8. Specify the following settings under the *Swept Variable(s)* tab:
   a. Select **Single variable** and **IWavePhi** from the two drop-down menus at the top of the tab.
   b. *Start*: **0 deg**
   c. *Stop*: **175 deg**
   d. *Steps*: **35**



> **Note:**
>
> This setup produces an animation with 5° increments of the incident phi angle, between 0° and 175° inclusively, and 36 frames total. The field overlay at 180° would be the same as at 0°. So the 180° angle is omitted from the animation intentionally. The absence of the repeated frame produces smooth playback when the animation loops back from the end to the beginning and continues to play.

9. Click **OK**.

   The frames are computed, the *Animation* dialog box appears, and the animation begins playing.

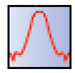
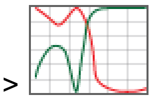


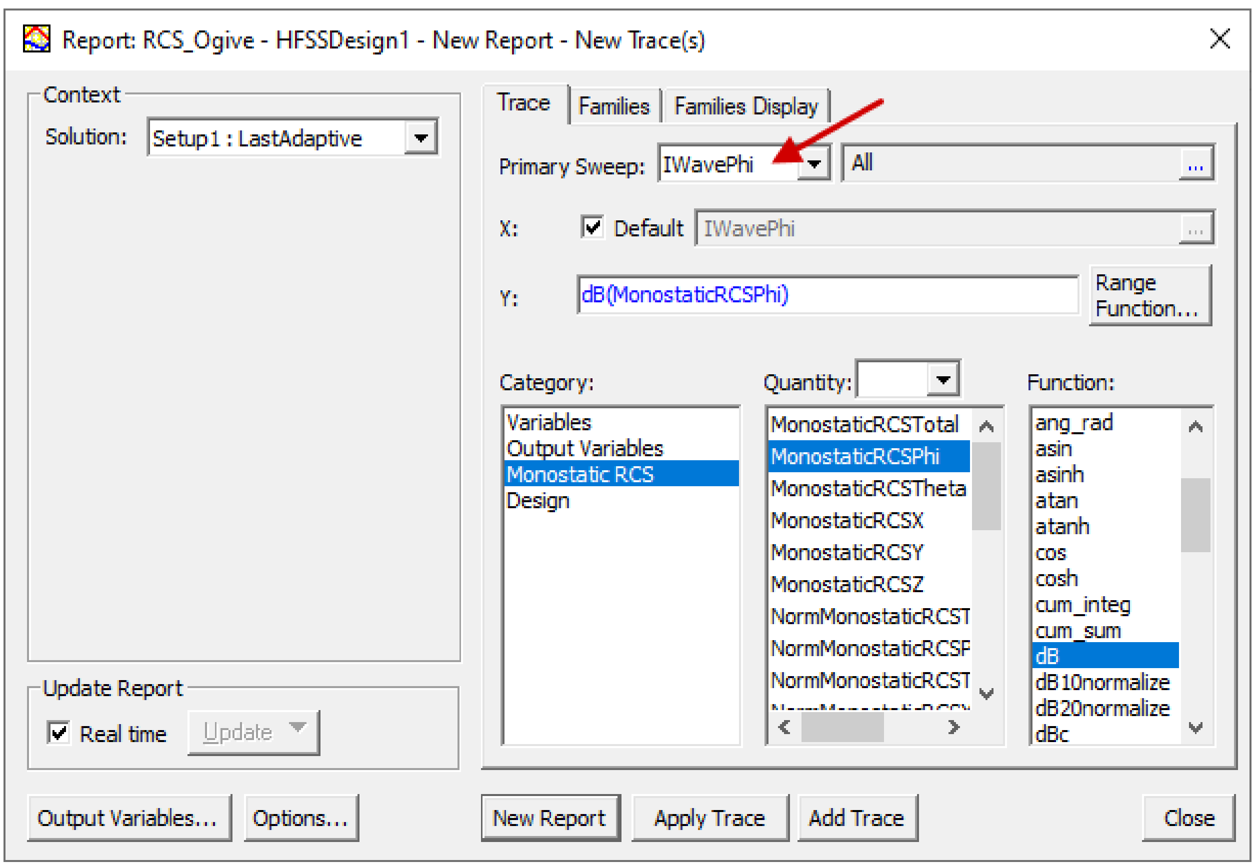
> **Note:**
>
> The maximum amplitude in the plot legend has increased to 0.0096, whereas it was 0.0059 for the static overlay. For IWavePhi values other than 0°, the peak surface current magnitude is greater than the value shown for the static overlay, which was based on IWavePhi = 0°. The legend scale is automatically adjusted to cover the range of results for the full sweep.

10. Use the controls in the *Animation* dialog box to stop, restart, reverse, or change the playing speed of the animation.
11. Click **Close** to stop the animation and dismiss the dialog box when you're done viewing the animated current results.

8 - Plot the RCS

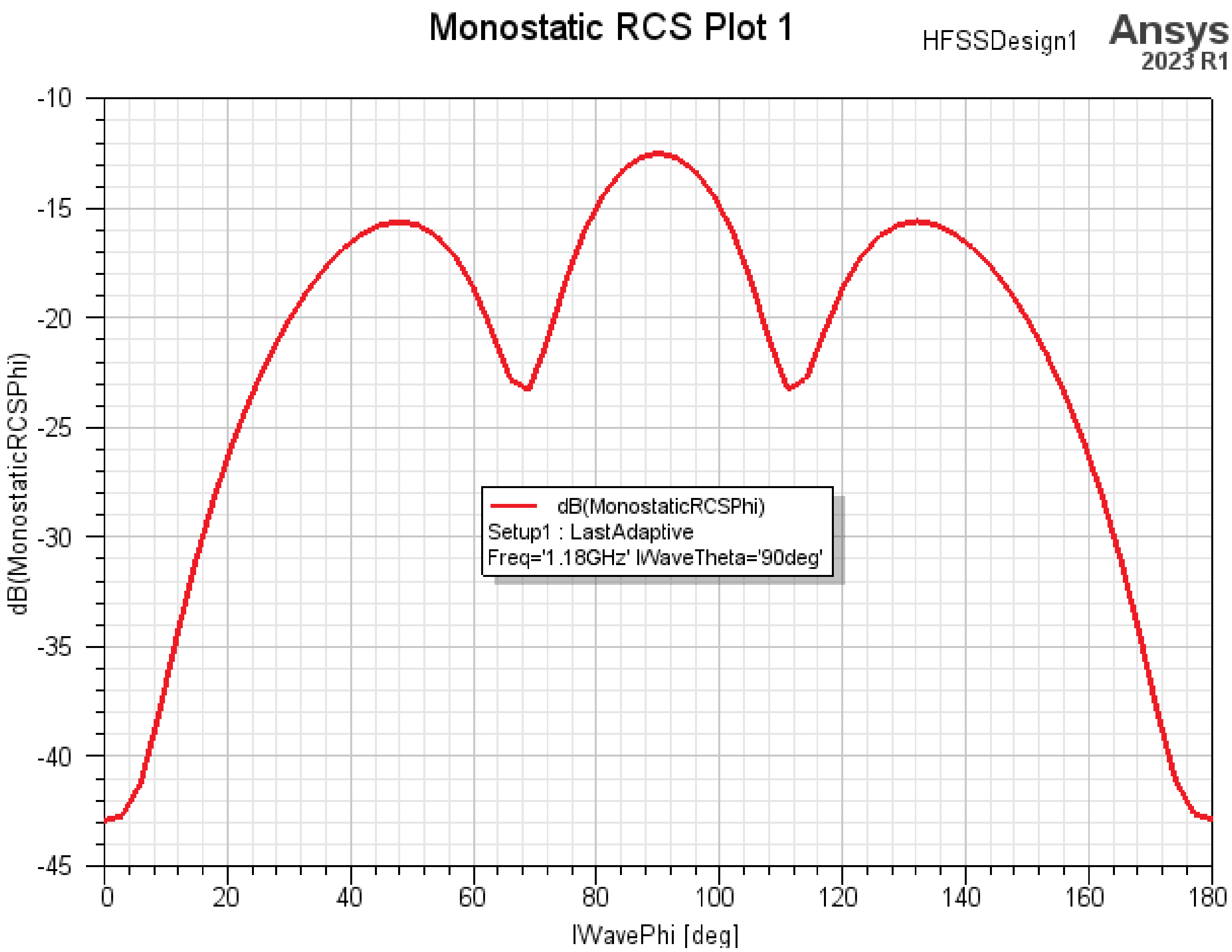
To plot monostatic RCS results in an HFSSdesign, you do not have to define an infinite sphere far field setup. For the monostatic RCS, you will be plotting against the incident phi angle (*IWavePhi*). Create the plot as follows:

1. Access the Report dialog box using one of the following two methods:
   - On the **Results** ribbon tab, click  **Monostatic RCS Report** >  **2D**.
   - Right-click **Results** in the Project Manager and choose **Create Monostatic RCS Report** > **Rectangular Plot** from the shortcut menu.
2. In the **Trace** tab of the *Report* dialog box, specify the following settings:
   a. *Primary Sweep*: **IWavePhi** (to plot the RCS vs. the incident phi angles)
   b. *Category*: **Monostatic RCS**
   c. *Quantity*: **MonostaticRCSPhi**
   d. *Function*: **dB**



3. Click **New Report** and **Close**.
4. Click and drag the **Curve Info** legend box to reposition it as desired.

   The resulting plot should resemble the following image:



   This plot is the HH monostatic RCS pattern for the standard ogive at 1.18GHz.

5. **Save** your project.

9 - Optionally, Restore Current View Orientations

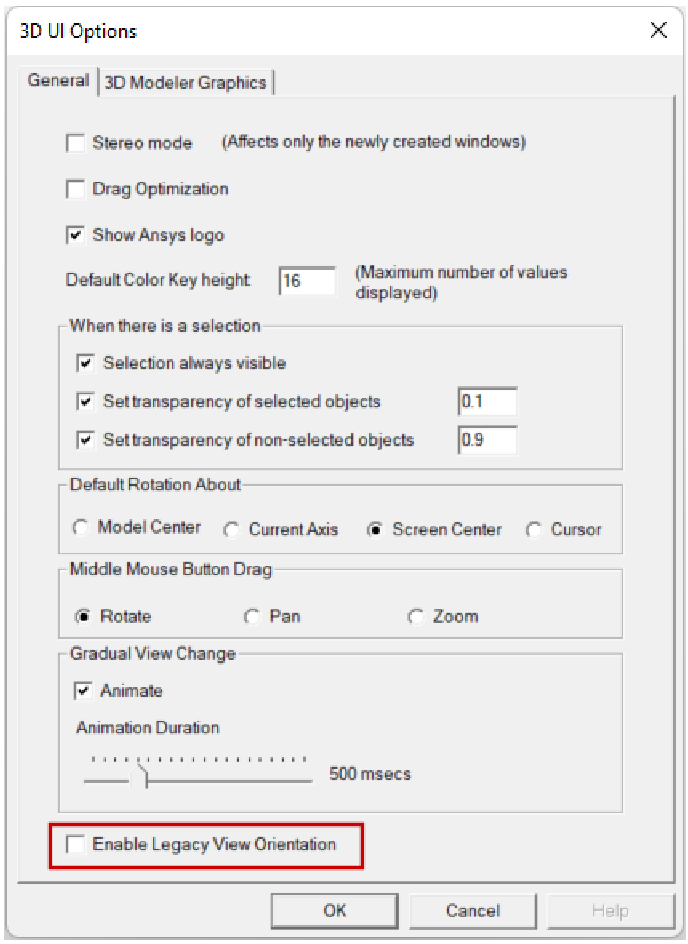
You have completed this getting started guide.

If you prefer to use the new view orientations implemented in version 2024 R1 of the Ansys Electronics Desktop application, clear the *Use Legacy View Orientation* option as follows:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears.

2. Ensure that **Enable Legacy View Orientation** is cleared:

3. Click **OK**.

The settings in the 3D UI Options dialog box are global. Your choice is retained for all future program sessions, projects, and design types that use the 3D Modeler or that produce 3D plots of results.

You can now save and close this project.