霍莱沃系统
Hollywave

RDSim

3D Electromagnetic Simulation Software

User Guide

2023 -02
Shanghai Hollywave Electronic System
Technology Co., Ltd.

CONTENTS

1 Home Module 1

1.1 New 1
1.2 Save 1
1.3 Open 2
1.4 Import 2
1.5 Solver 2
1.6 Variables 3
1.7 History 5
1.8 Log 6
1.9 Help 7
1.10 Settings 7
1.11 Script 8

2 Model Module 10

2.1 Unit 10
2.2 View 11
2.2.1 Select Settings 11
2.2.2 Modeling View Settings 11
2.3 Coordinate System 11
2.4 Line 12
2.4.1 Line 13
2.4.2 Polyline 13
2.4.3 Helix 14
2.4.4 Arc 15
2.4.5 Bezier 16
2.4.6 Parabola 17
2.4.7 Hyperbola 18
2.5 Face 19
2.5.1 Triangle 19
2.5.2 Rectangular 20
2.5.3 Circles 21

2.5.4 Ellipse 22
2.5.5 Polygon 23
2.5.6 Paraboloid 24
2.6 Body 25
2.6.1 Box 25
2.6.2 Cylinder 26
2.6.3 Sphere 27
2.6.4 Ellipsoid 28
2.6.5 Four-Sided Platform 29
2.6.6 Cone 30
2.6.7 Torus 31
2.7 Tool 32
2.7.1 Transform 32
2.7.2 Copy 36
2.7.3 Boolean 39
2.7.4 Sweep 44
2.7.5 Connect 45
2.7.6 Cover Line To Surface 46
2.8 Array 47
2.9 Pick 49
2.10 Bend 50
3 Material Setting Module 51
3.1 Material library 51
3.2 Material search 52
3.3 New material 53
3.3.1 New dielectric material 53
3.3.2 New metal material 57
3.3.3 New Anisotropic - Dielectric Material 60
3.3.4 New Anisotropic - Metal Material 64
3.3.5 New Impedance Condition Material 68
3.4 Surface Model Material Settings 70
3.5 Body model material settings 74
3.6 Material settings for faces in body models 78

3.7 Coating material 82
3.7.1 Coating material settings on model surfaces 82
3.7.2 Coating material settings for volume model surfaces 85
4 View Module 88
4.1 Orient 88
4.1.1 Orient 88
4.1.2 Move/Rotate 88
4.1.3 Zoom 89
4.2 Tool 89
4.2.1 Show 89
4.2.2 Wireframe 90
4.2.3 Grid 91
4.2.4 Coordinate System 92
4.2.5 Transparency 92
4.3 Measure 94
4.3.1 Measure Length 94
4.3.2 Measuring Angle 96
4.4 Cutting Plane 98
4.5 Screen 100
4.5.1 Screenshot 100
4.5.2 Window 100
5 Excitation and Monitor module 101
5.1 Frequency 101
5.1.1 Single frequency 101
5.1.2 Linear interval 102
5.1.3 Log interval 103
5.1.4 Discrete point 104
5.2 Port 105
5.2.1 Connection port 105
5.2.2 Waveguide port 108
5.2.3 Lumped port 114
5.3 Ideal source 117

5.3.1 Plane wave 117
5.3.2 Electric dipole 119
5.3.3 Magnetic dipole 121
5.4 Field source 123
5.4.1 Near Field 123
5.4.2 Far Field 126
5.4.3 Spherical Wave 127
5.5 Synthesis 129
5.6 Components 131
5.6.1 Line Components 131
5.6.2 Boundary Components 135
5.7 Monitor 140
5.7.1 Current result settings 140
5.7.2 Near Field Results Settings 143
5.7.3 Far Field Result Settings 145

6 Meshing Module 151

6.1 Create mesh 151
6.2 Mesh Check 153
6.2.1 Intersecting Models 153
6.2.2 Intersecting Triangles 154
6.2.3 Distorted Elements 155
6.2.4 Oversized Elements 156
6.3 Refinement 158
6.3.1 Point Refinement 158
6.3.2 Line Refinement 159
6.3.3 Local Mesh 161
6.4 Mesh Information 163
6.5 Mesh Cleanup 164

7 Simulation module 165

7.1 Validate 165
7.2 Parameter Sweep 165
7.3 Algorithm settings 167

7.3.1 MOM Settings 167
7.3.2 MLFMA algorithm settings 167
7.3.3 SBR+ algorithm setting 170
7.4 Run 171
7.5 Options 172
7.5.1 Hardware options 172
7.5.2 HPC 173
7.5.3 Remote 173
8 Result module 175
8.1 Import/Export 175
8.1.1 Import Data 175
8.1.2 Export Data 175
8.1.3 Export Plot 175
8.2 Export Result Settings 176
8.2.1 Near Field 176
8.2.2 Far Field 176
8.3 Result 177
8.3.1 Model parameters 177
8.3.2 Electric current 179
8.3.3 Near field 182
8.3.4 Far field 190

1 Home Module

The **Home** module includes **File**, **Setup**, **Mesh**, **Optimization**, **Solver**, **Edit**, **Help**, **Default Settings** and **Script**.

The **File** section includes **New**, **Save**, **Save as**, **Open**, **Import** and **Export** options. The **Setup** section includes **Unit** options. The **Mesh** section includes **Create mesh**, **Mesh Settings** and **Mesh Check** options. The Optimization section includes **Parameter Sweep**, **Sensitivity** and **Optimization** options. The **Solver** section includes **Validate**, **Solver Setup**, **Run** and **Solution Method** options. The **Edit** section includes **Variables**, **History** and **Log** options. The **Help** section includes **Manual** and **Examples**. The **Default Settings** is used to set the initial setting when opening software.

1.1 New

Click **New** to create a new project folder.

The steps to create a new project folder are as follows:

1、 Click the **New** button.

2、 Select path for the project folder in the window and enter the project name.

3、 Click **Save** to finish.

1.2 Save

Click **Save** to save the current project.

Click **Save as** to save the current project to a new path.

The steps to save as are as follows:

1、 Click **Save** button.

2、 Select path for the project in the pop-up window and enter the project name.

3、 Click confirm to save.

1.3 Open

Click **Open** to open a previous project.

The steps to open are as follows:

1、 Click **Open** button.

2、 In the pop-up window, open the target project folder.

3、 Select **entry.rdsim**

4、 Click open.

1.4 Import

Click **Import** to import 3D CAD models and mesh models.

The steps to import a 3D CAD model are as follows:

1、 Click **Import** button.

2、 Click **3D CAD.**

3、 Select a file type (.stp or.brep).

4、 In the pop-up window, select a 3D CAD model file to be imported.

5、 Click open.

The operation steps of importing a mesh model are as follows:

1、 Click **Import** button.

2、 Click **Mesh**.

3、 Select a file type (.stl or.nas).

4、 In the pop-up window, select a mesh model file to be imported.

5、 Click open.

1.5 Solver

The setting of the solution method is used to switch between **MoM** solution and

C**haracteristic Model Analysis(CMA)** solution. In this user guide, we set the solution method as **MoM**.

1.6 Variables

The **Variable** option is used to set variables, user can add and delete variables. Variables can be applied in modeling, frequency, parameter sweep and results.

The expression of a variable can be a value or an existing variable name. Once an expression is entered, the value of the variable will appear in **value**. If an an undefined variable is entered in expression, errors will be reported. The variable name only expresses numerical value without units.

The steps to add a variable are:

1、 Method 1:

(1) Click **Variables** button.

(2) In the variable pop-up window, click **Input variable**.

(3) Input variable name, expression and description (non-essential).

(4) Click **Confirm** to complete.

(5) In the variable manager area, the variable name, expression, value and description of the new variable will appear.

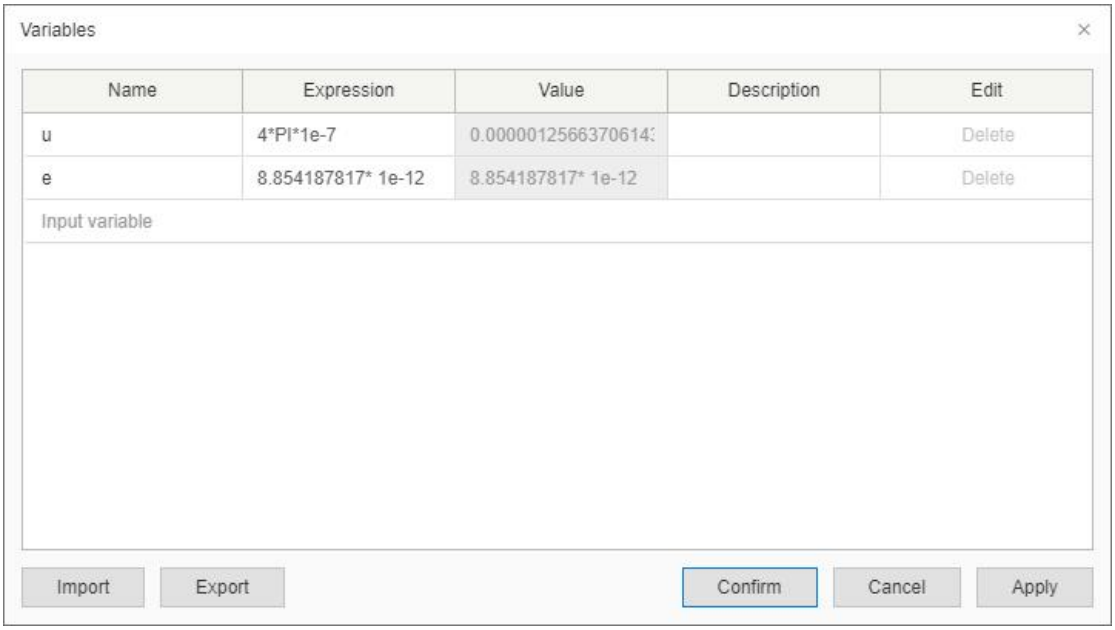


Figure 1 New variables

Figure 2 New variables—variables manager area

2、 Method 2:

(1) Click **Input variable** in the variable manager area.

(2) In the input box, enter variable name, expression and description (non-essential).

(3) Click ☑ button to complete.

(4) The variable name, expression, value and description will appear in variable manager area.

The steps to delete a variable are:

1、 Method 1:

(1) Click **Variables** button.

(2) In the variable window, click the selection box of a variable.

(3) Click **Delete** to finish.

(4) In the variable manager area, the variable name, expression, value and description of the variable will be deleted.

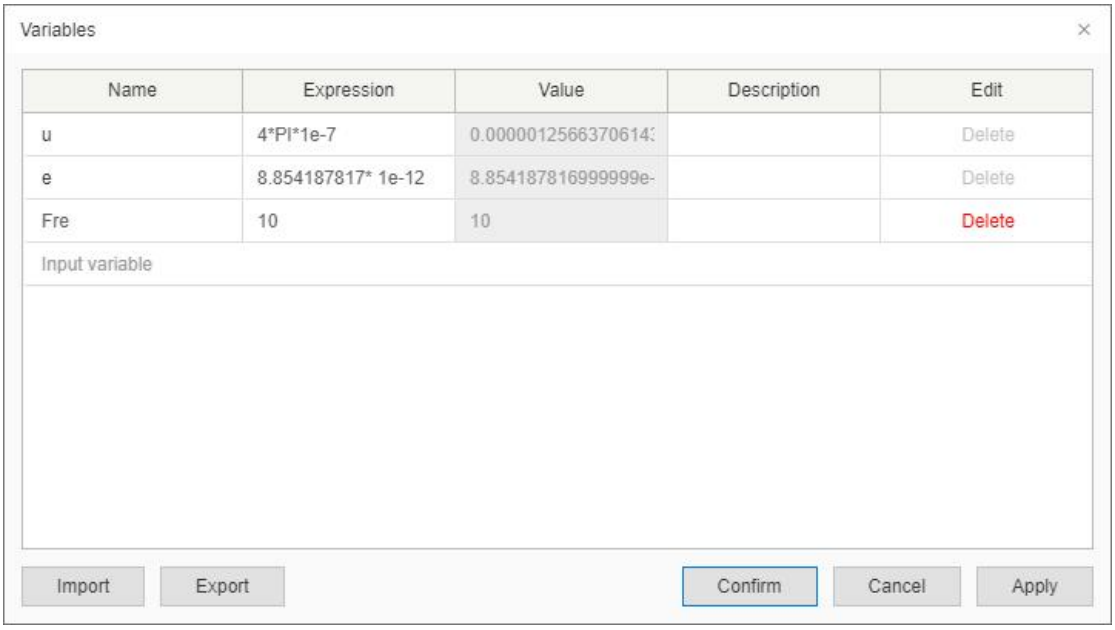


Figure 3 Delete variable

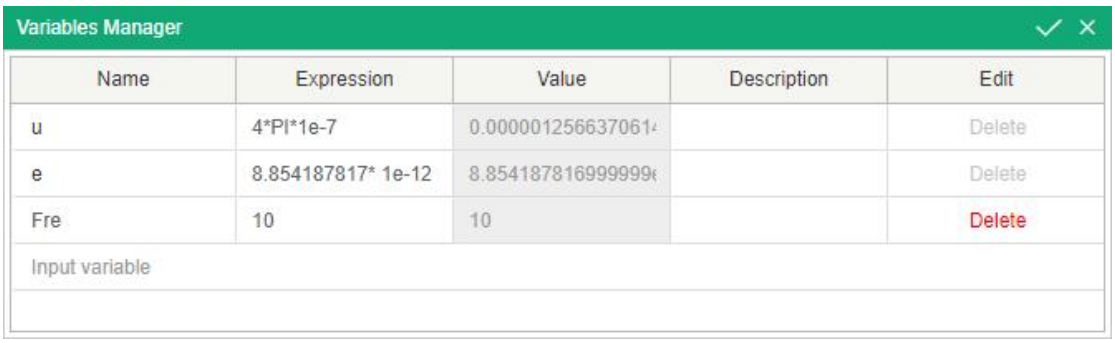


Figure 4 Delete variable—variables manager

2、 Method 2:

(1) Click the selection of a variable in the variable manager area.

(2) Click **Delete** in the variable manager area.

1.7 History

History records all operations that have been processed in the project, including modeling, material settings, frequency settings, excitation settings, result settings and other operation processes. History contains functions of **Run to**, **Step**, **Continue**, **Delete**, **Edit**, and **Delete all.**

Run to runs all operations from the first step of the project to the selected step. **Step** performs a single operation in the history. **Continue** runs history from the

current step to the selected step. **Delete** deletes the selected step. **Edit** modifies the selected step. **Delete all** clears all history.

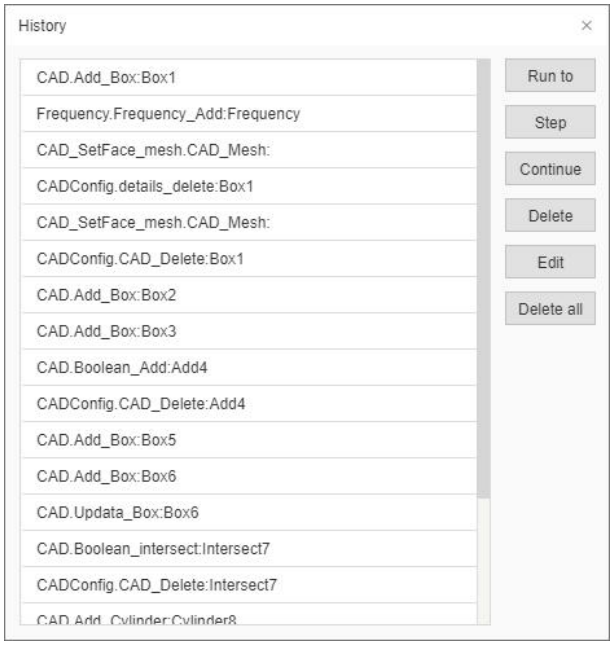


Figure 5 History

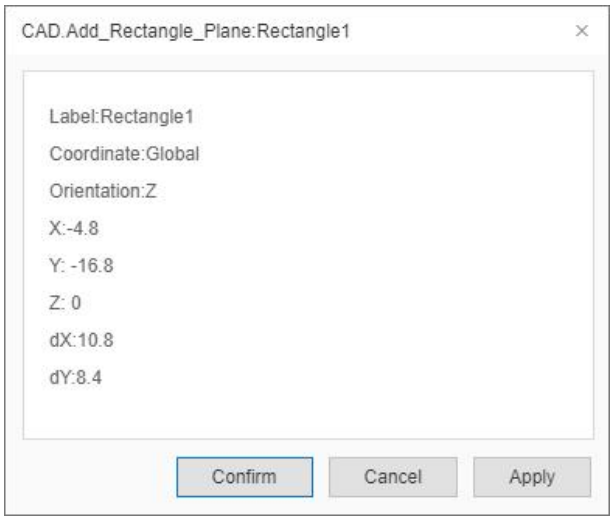


Figure 6 History-edit

1.8 Log

View detailed information of algorithm output of the previous simulation in **Log**.

Figure 7 Log

1.9 Help

Click **Help** to view **User Manual** and **Examples**.

1.10 Settings

Settings include **Model s**ettings, **Show** settings, **Mesh** settings, **Shortcut Keys** settings, **Monitor** settings, **Hardware Options** and **Result attribute** settings. Users can edit default settings of the corresponding parts independently.

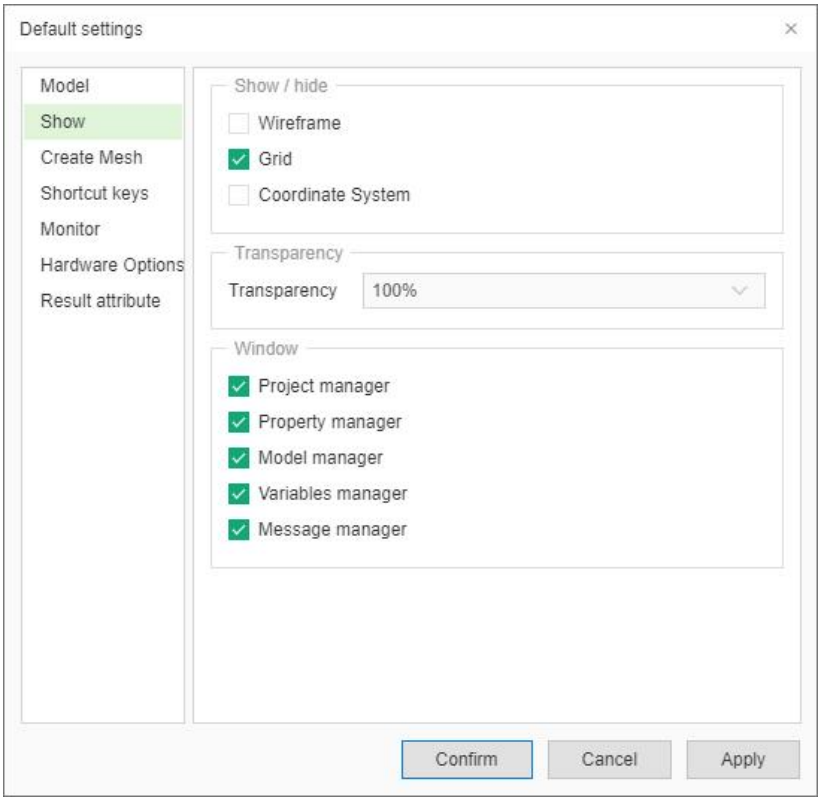


Figure 8 Settings

1.11 Script

Click **Script** to quickly realize specific operations according to the input script.

The steps to run the script are:

1、 Enter script commands for desired operations in the script box.

2、 Click **Run** to run the script command operations in sequence.

3、 Click **Reset** to reset the script.

4、 Click **Import** to select a script file to import and run.

5、 Click **Export** to export the current script and save to a local path.

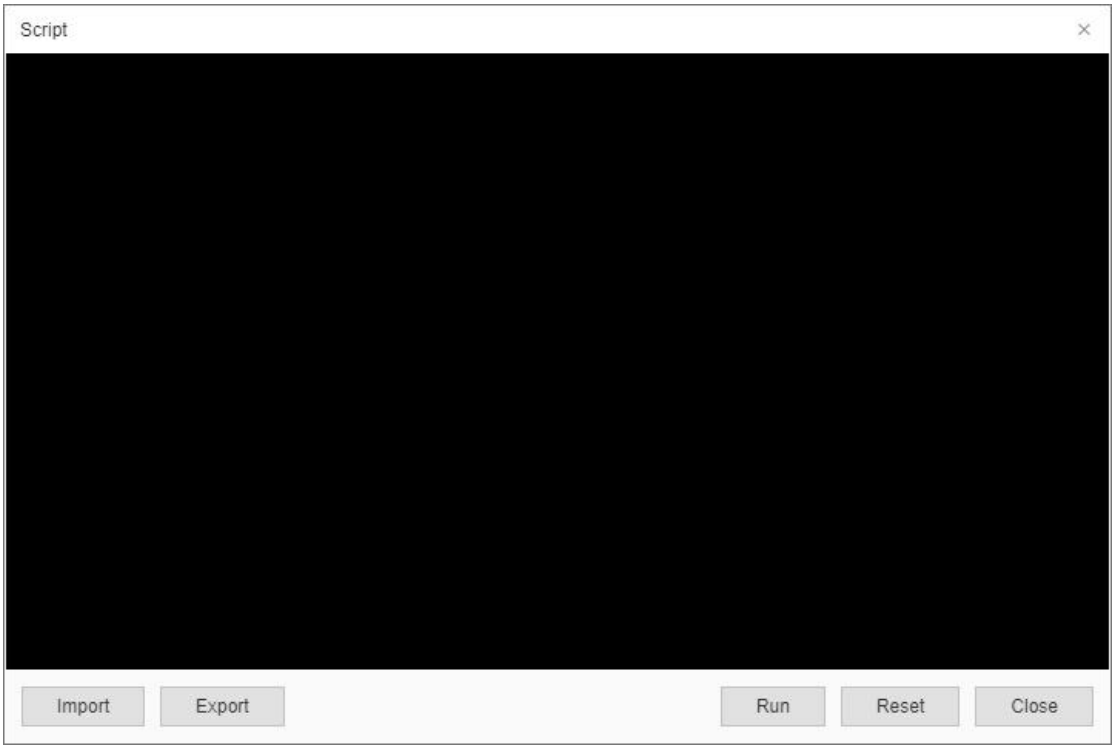


Figure 9 Script

2 Model Module

Model modules define model units, model work spaces, etc. In **model** module, users can establish basic models and apply Boolean operations to models.

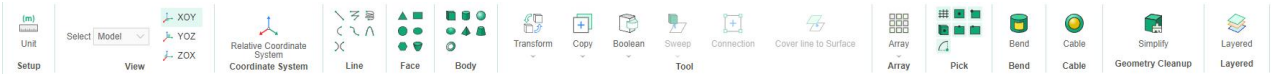


Figure 10 Modeling module function section

2.1 Unit

Unit include **Dimension** units, **Frequency** units, **Temperature** units, **Current** units, **Resistance** units, **Voltage** units, **Conductance** units, **Capacitance** units, **Inductance** units.

Dimension units apply to all models in the project.

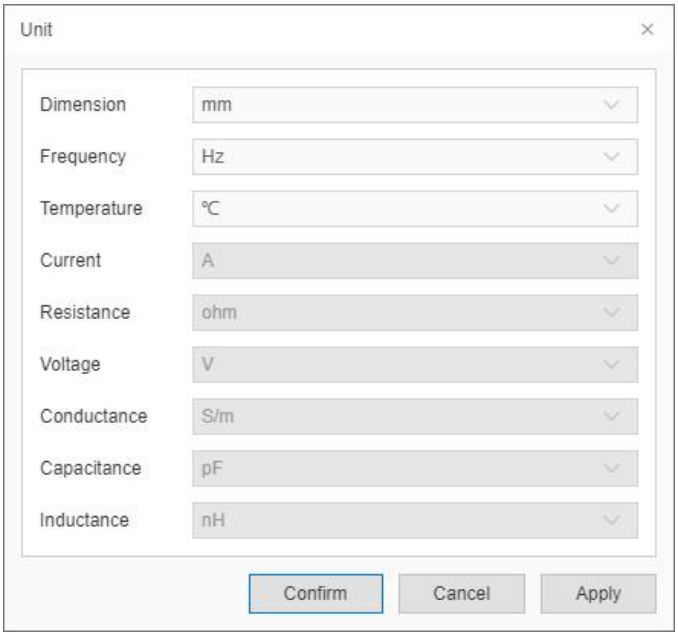


Figure 11 Unit

The steps to modify units are:

1、 Select the **Model** tab in the function area.

2、 Click the **Unit** button.

3、 Select desired units, and click **Confirm** to complete the unit setting (default units are mm for dimension, Hz for frequency, °C for temperature).

2.2 View

View settings includes model **select** settings and model **view** settings.

2.2.1 Select Settings

Select Settings defines ways to select models, including select models, bodies, surfaces, lines and points.

The steps of select settings are:

1、 Click the **Select** button.

2、 Select model, body, surface, line and vertex.

3、 Move the mouse to the view area and select respective models.

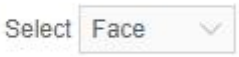


Figure 12 Select settings

2.2.2 Modeling View Settings

Modeling View Settings set the plane where the model bottom is located. If **XOY** plane is selected, the model bottom is parallel to XOY plane. if **XOZ** plane is selected, the model bottom is parallel to XOZ plane. if **YOZ** plane is selected, the model bottom is parallel to YOZ plane.

Click **XOY** or **YOZ** or **ZOX** button to set the model view.

Figure 13 Modeling view settings

2.3 Coordinate System

The **Coordinate System** defines the relative coordinate systems.

The steps to build relative coordinate system are:

1、 Click the **Coordinate System** button.

2、 In the viewing area, double click the left mouse to select a suitable point.

3、 The property interface of the relative coordinate system will pop up in the viewing area. Enter label name, starting point, x-axis unit vector, y-axis unit vector, and z-axis unit vector of the relative coordinate system, and click **Confirm** to finish.

Figure 14 Coordinate system

2.4 Line

Line includes **Line**, **Polyline**, **Helix Curve**, **Arc**, **Bezier**, **Parabola** and **Hyperbola**.

Button	Label Name	Function
	Line	Create a line model
	Polyline	Create a polyline model
	Helix Curve	Create a spiral model
	Arc	Create an arc model
	Bezier curve	Create a bezier curve model
	Parabola	Create a parabolic model
	Hyperbola	Create a hyperbolic model

2.4.1 Line

The steps to create a straight line model are:

1、 In the **Line** model column of the **Model** section, click the **Line** button.

2、 Double click in the view area to establish the starting point of the line.

3、 Then double-click the left mouse button to establish the end point of the line.

4、 In the pop-up window, enter the model label name, and modify the starting point and end point coordinates.

5、 Finally, click **Confirm** to finish.

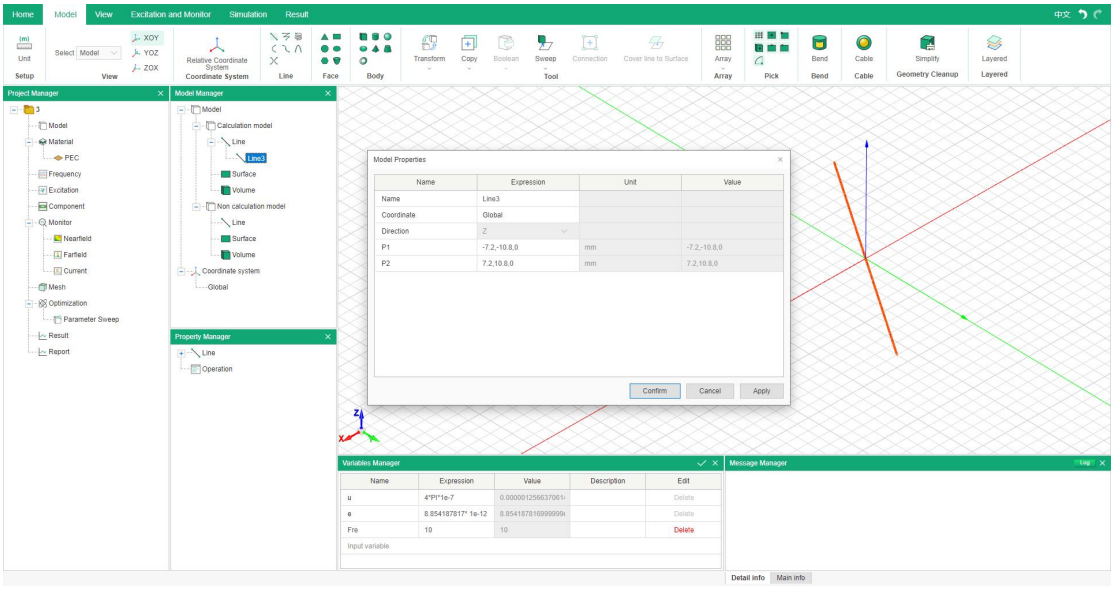


Figure 15 Line

2.4.2 Polyline

The steps to create a polyline model are:

1、 In the **Line** model column of the **Model** section,click the **Polyline** button.

2、 Double-click the left mouse button in the view area to establish the starting point of the polyline.

3、 Then click the mouse in turn to determine other vertices.

4、 Finally, double-click the left mouse button to establish the end point of the polyline.

5、 In the pop-up window, enter the model label name and the coordinates of each vertex.

6、 Finally, click **Confirm** to finish.

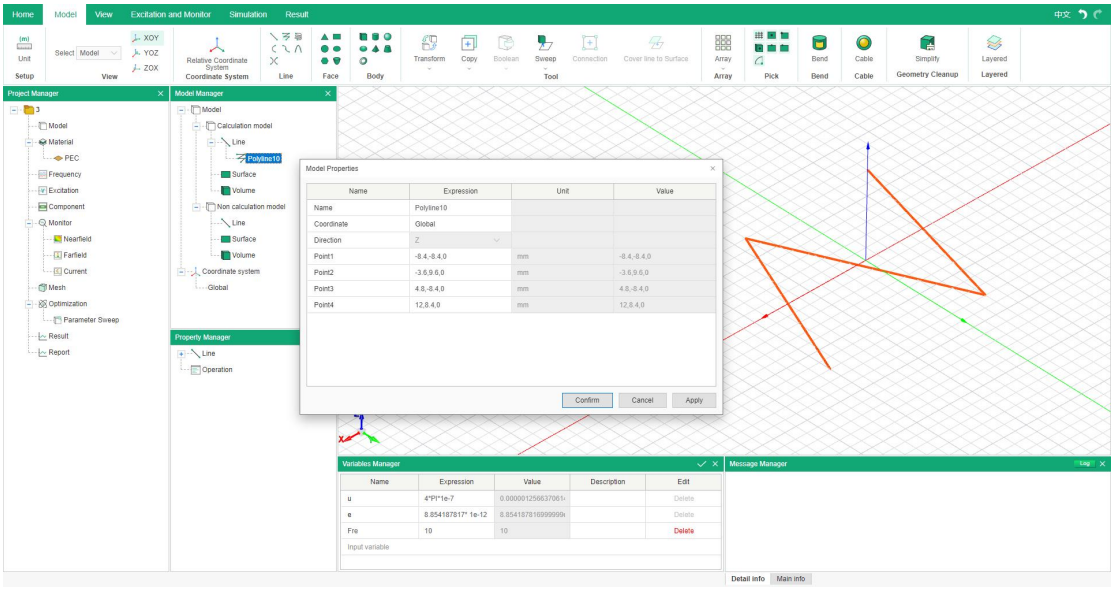


Figure 16 Polyline

2.4.3 Helix

The steps to create a helix model are:

1、 In the **Line** model column of the **Model** section, click the **Helix** button.

2、 Double click in the view area to establish the starting point of the spiral.

3、 Then double-click the left mouse button to determine the end point of the spiral.

4、 In the pop-up window, enter the model label name, starting point,

height, bottom/top radius and number of turns.

5、 Finally click **Confirm** to finish.

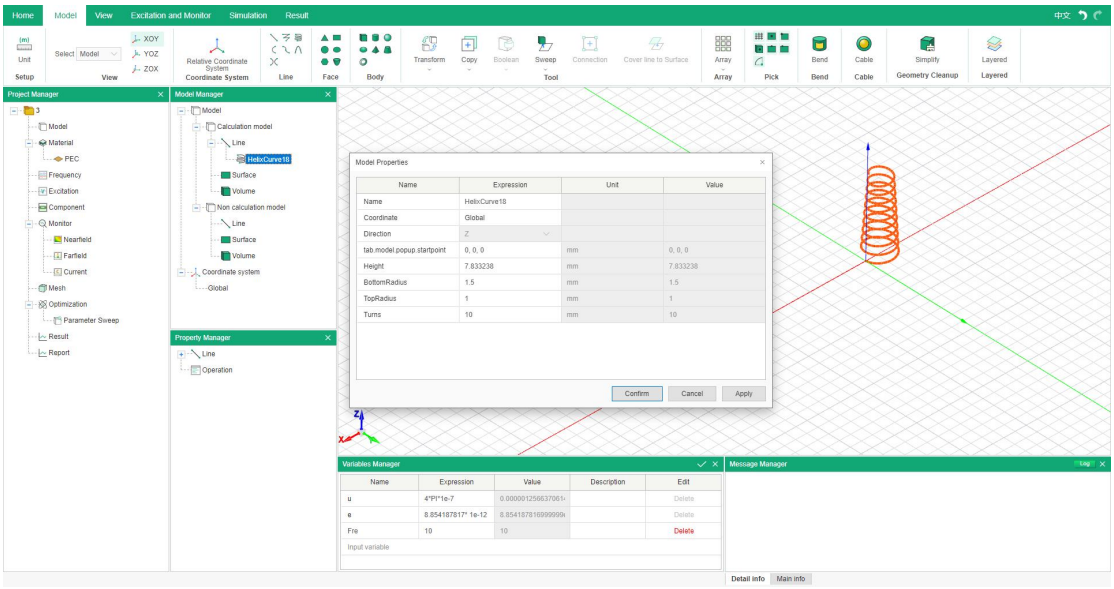


Figure 17 Helix

2.4.4 Arc

The steps to create the arc are:

1、 In the **Line** model column of the **Model** section, click the **Arc** button.

2、 Double-click in the view area to determine the center of the arc.

3、 Then double click the left mouse button to determine the radius of the arc.

4、 Enter the model label name, center, X/Y axis radius, starting Angle and ending Angle in the pop-up window.

5、 Finally, click **Confirm** to finish.

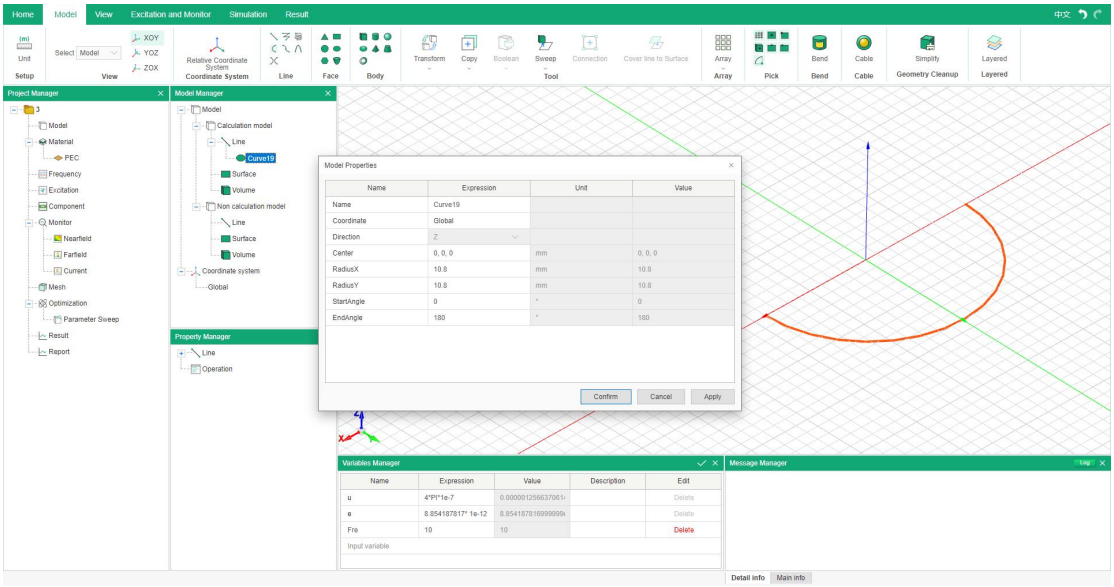


Figure 18 Arc

2.4.5 Bezier

The steps to create a Bezier curve model are:

1、 In the **Line** model column of the **Model** section, click the **Bezier Curve** button.

2、 Double-click in the view area to determine the starting point of the Bezier curve.

3、 Then click the mouse in turn to determine other vertices.

4、 Then double-click the left mouse button to determine the end point of the Bezier curve.

5、 In the pop-up window, enter the model label name and the position of each vertex of the Bezier curve.

6、 Finally, click **Confirm** to finish.

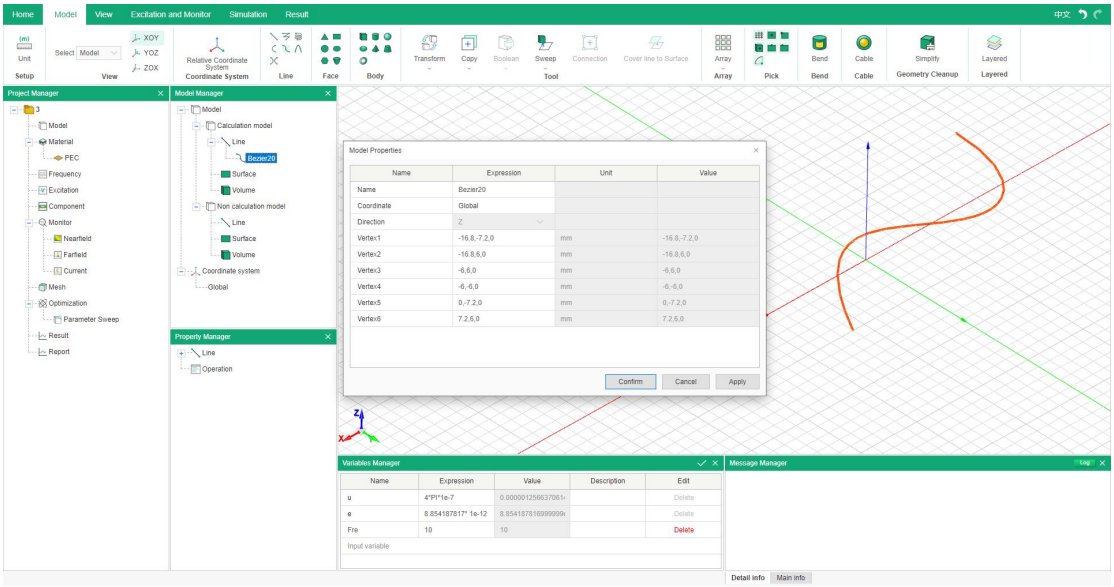


Figure 19 Bezier curve

2.4.6 Parabola

The steps to create a parabolic model are:

1、 In the **Line** model column of the **Model** section,click the **Parabola** button.

2、 In the pop-up window, enter the model label name, parabola vertex, focal length and radius.

3、 Finally, click **Confirm** to finish.

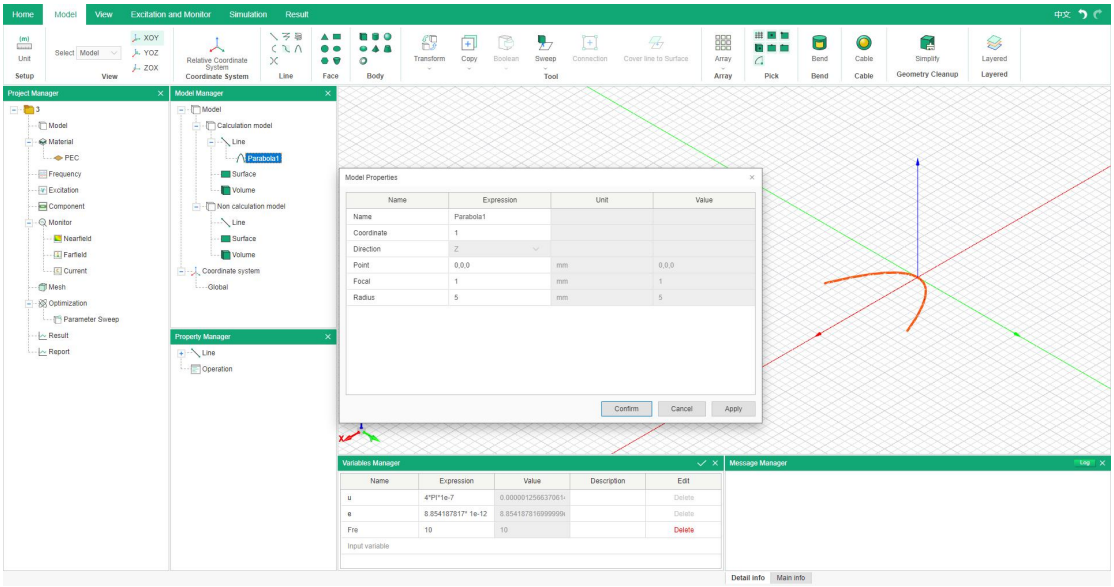


Figure 20 Parabola

2.4.7 Hyperbola

The steps to create a hyperbolic model are:

1、 In the **Line** model column of the **Model** section, click the **Hyperbola** button.

2、 In the pop-up window, enter the model label name, main axis distance, auxiliary axis distance and radius.

3、 Finally click **Confirm** to finish.

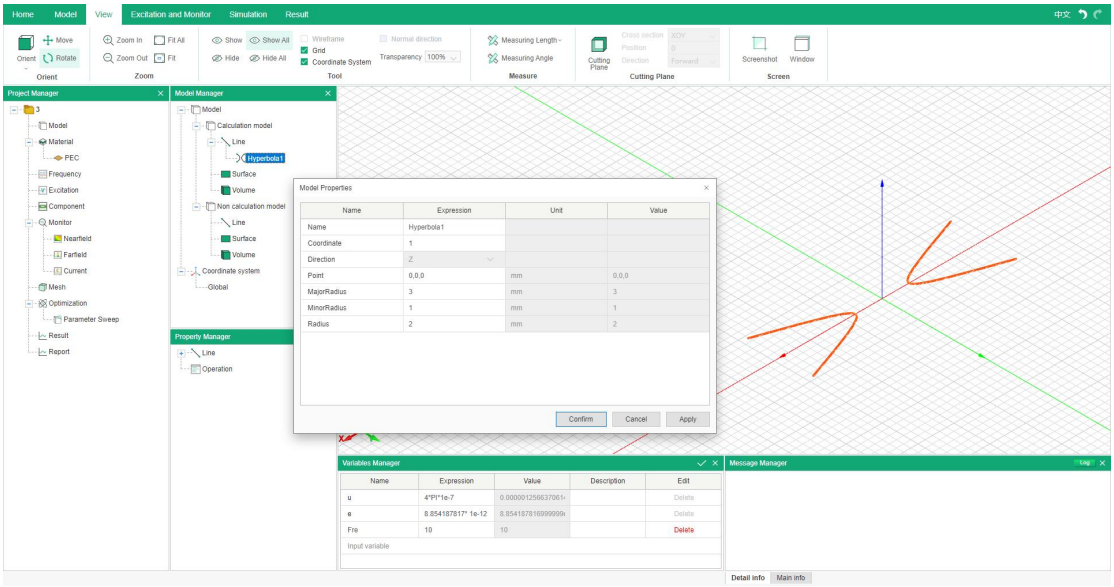


Figure 21 Hyperbola

2.5 Face

Face includes **Triangle**, **Rectangle**, **Circle**, **Ellipse**, **Polygon**, and **Paraboloid**.

Button	Label name	Function
	Triangle	Create a triangle model
	Rectangle	Create a rectangular model
	Circle	Create a circular model
	Ellipse	Create an Ellipse Model
	Polygon	Create a regular polygon model
	Paraboloid	Create a parabolic model

2.5.1 Triangle

The steps to create a triangle model are:

1、 In the **Face** model column of the **Model** section, click **Triangle** button.

2、 Double-click the left mouse button in the view area to determine a

vertex of the triangle.

3、 Move the mouse and click the left mouse button to determine the second vertex of the triangle.

4、 Move the mouse, double- click the left mouse button to determine the last vertex of the triangle.

5、 In the pop-up window, enter the model label name and the coordinates of each vertex.

6、 Finally, click **Confirm** to finish.

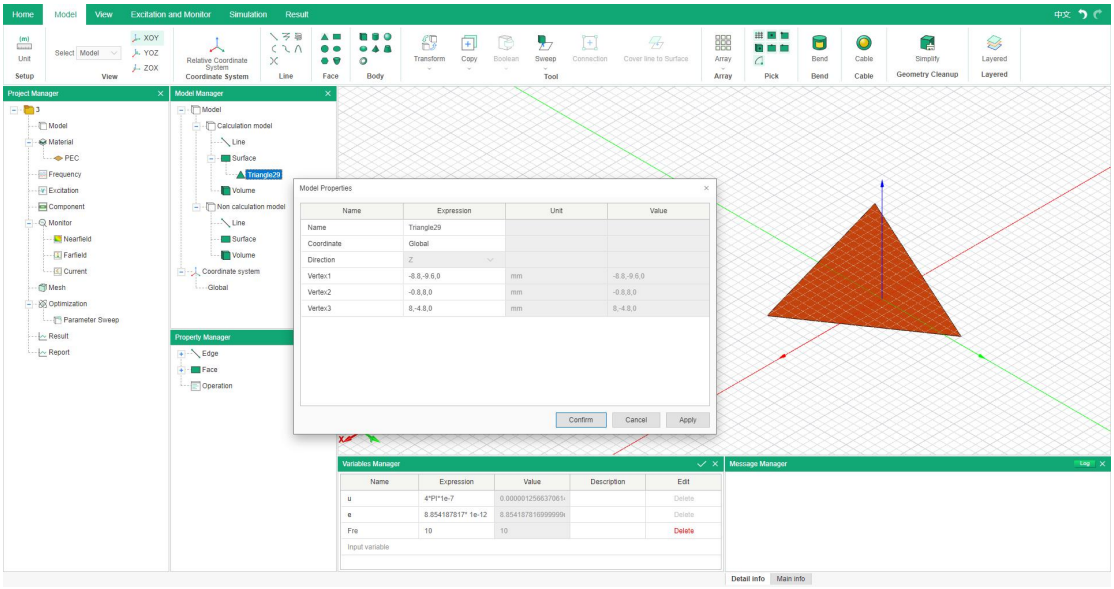


Figure 22 Triangle

2.5.2 Rectangular

The steps to create a rectangular model are:

1、 In the **Face** model column of the **Model** section, click **Rectangle** button.

2、 Double-click the left mouse button in the viewing area to determine a vertex of the rectangle.

3、 Then move the mouse and double-click the left mouse button to determine another vertex of the rectangle.

4、 In the pop-up window, enter the model label name, vertex

coordinates, X-axis length, and Y-axis length.

5、 Finally, click **Confirm** to finish.

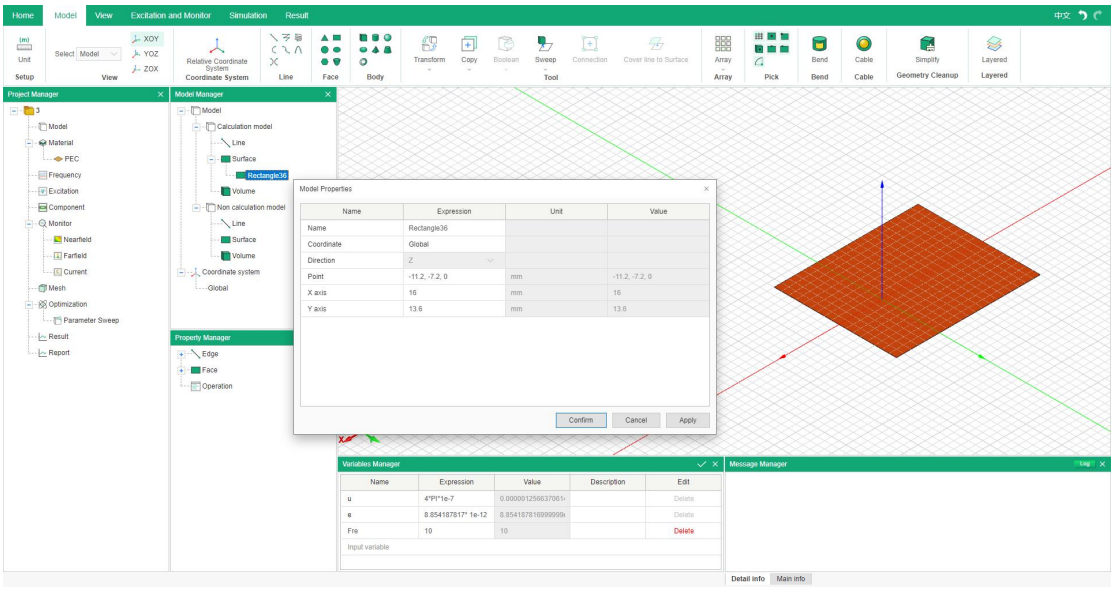


Figure 23 Rectangle

2.5.3 Circles

The steps to create a circular model are:

1、 In the **Face** model column of the **Model** section, click **Circle** button.

2、 Double-click the left mouse button in the view area to determine the center of the circle.

3、 Then move the mouse, double-click the left mouse button to determine the radius of the circle.

4、 In the pop-up window, enter the model label name, center position, and radius value.

5、 Finally click **Confirm** to finish.

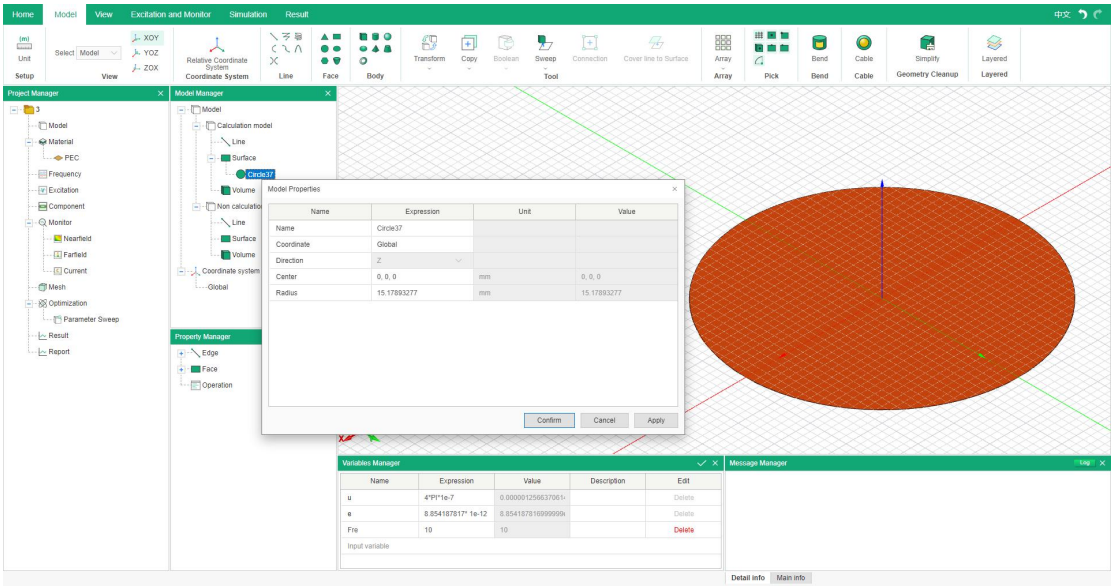


Figure 24 Circles

2.5.4 Ellipse

The steps to create an ellipse model are:

1、 In the **Face** model column of the **Model** section, click **Ellipse** button.

2、 Double-click the left mouse button in the viewing area to determine the center of the ellipse.

3、 Then move the mouse, double-click the left mouse button to determine the radius of the ellipse.

4、 In the pop-up window, enter the model label name, center position, and X/Y semi-axis length.

5、 Finally click **Confirm** to finish.

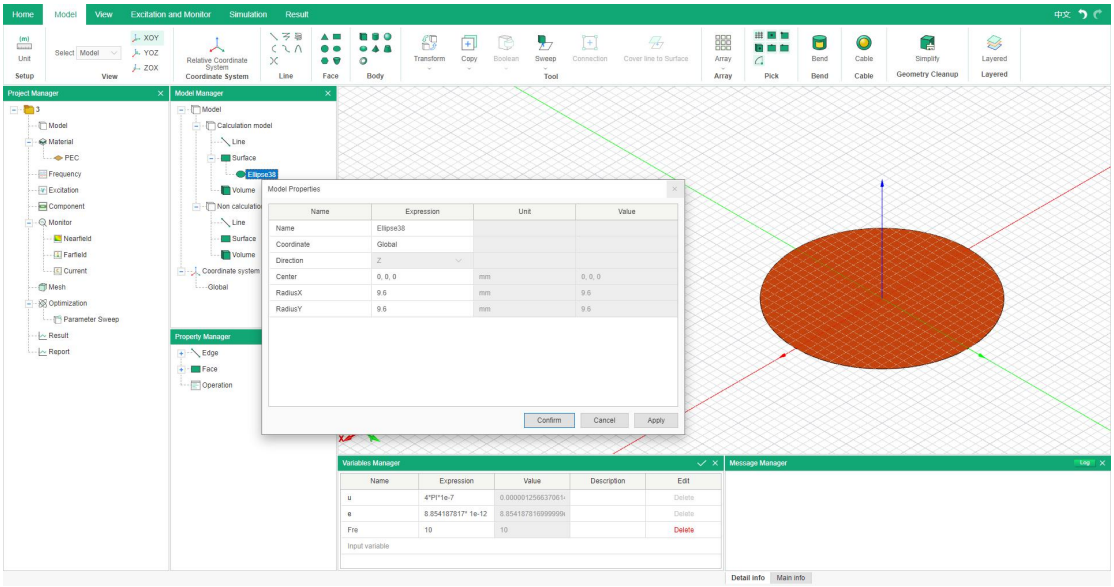


Figure 25 Ellipse

2.5.5 Polygon

The steps to create a polygonal model are:

1、 In the **Face** model column of the **Model** section, click **Polygon** button.

2、 Double-click the left mouse button in the view area to determine the first vertex of the polygon.

3、 Then click the mouse in turn to determine other vertices.

4、 Finally, double-click the left mouse button to determine the last vertex of the polygon.

5、 In the pop-up window, enter the model label name and the coordinates of each vertex.

6、 Finally click **Confirm** to finish.

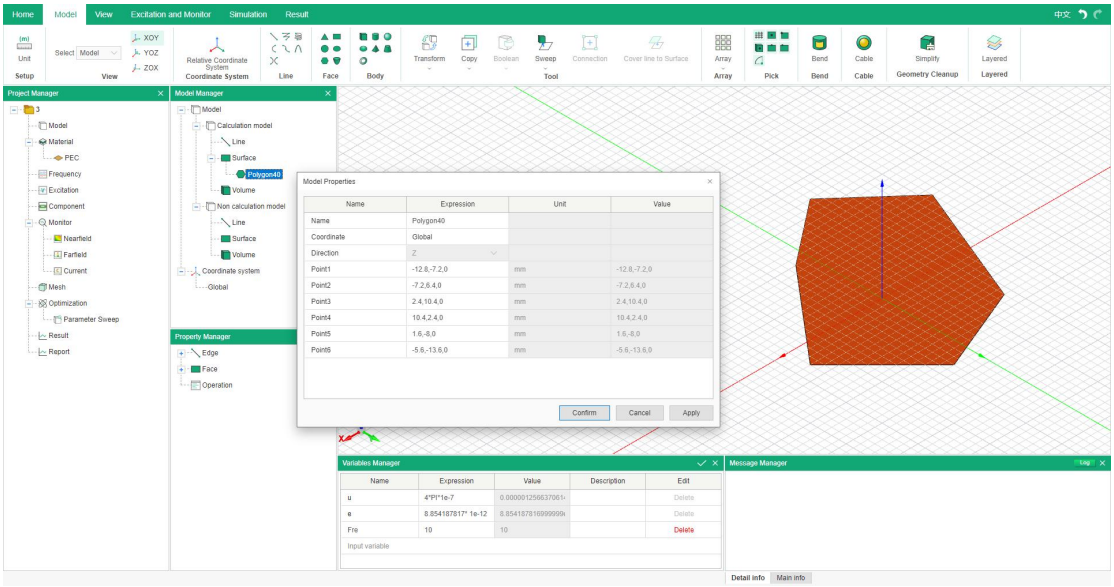


Figure 26 Polygon

2.5.6 Paraboloid

The steps to create a paraboloid model are:

1、 In the **Face** model column of the **Model** section, click **Paraboloid** button.

2、 In the pop-up window, enter the model label name, paraboloid vertex, focal length and radius.

3、 Finally, click **Confirm** to finish.

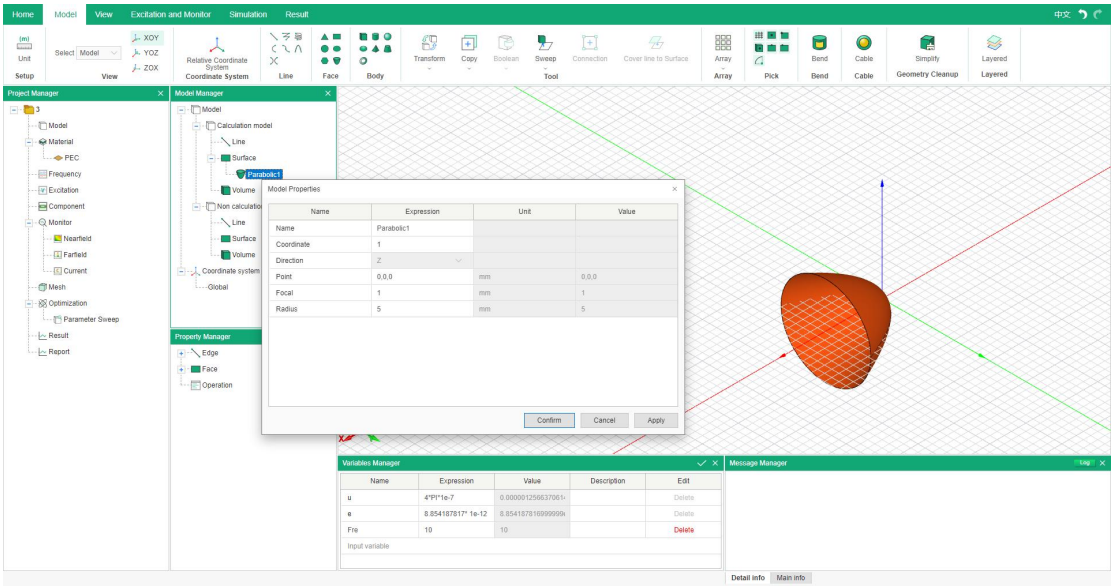


Figure 27 Paraboloid

2.6 Body

Body include **Cuboid**, **Cylinder**, **Sphere**, **Ellipsoid**, **Four-Sided Platform**, **Cone** and **Torus**.

Button	Label name	Function
	Box	Create a cuboid model
	Cylinder	Create a cylinder model
	Sphere	Create a sphere model
	Ellipsoid	Create an ellipsoid model
	Four-Sided Platform	Create a four-sided platform model
	Cones	Create a Cone Model
	Torus	Create a Torus model

2.6.1 Box

The steps to create a box model are:

1、 In the **Body** model column of the **Model** section, click the **Box**

button.

2、 Double-click the left mouse button in the view area to determine the position of the bottom vertex of the cuboid.

3、 Then move the mouse and click the left mouse button to determine the size of the bottom of the cuboid.

4、 Then move the mouse, double- click the left mouse button to determine the height of the cuboid.

5、 In the pop-up window, enter the model label name, vertex, and X/Y/Z axis length.

6、 Finally, click **Confirm** to finish.

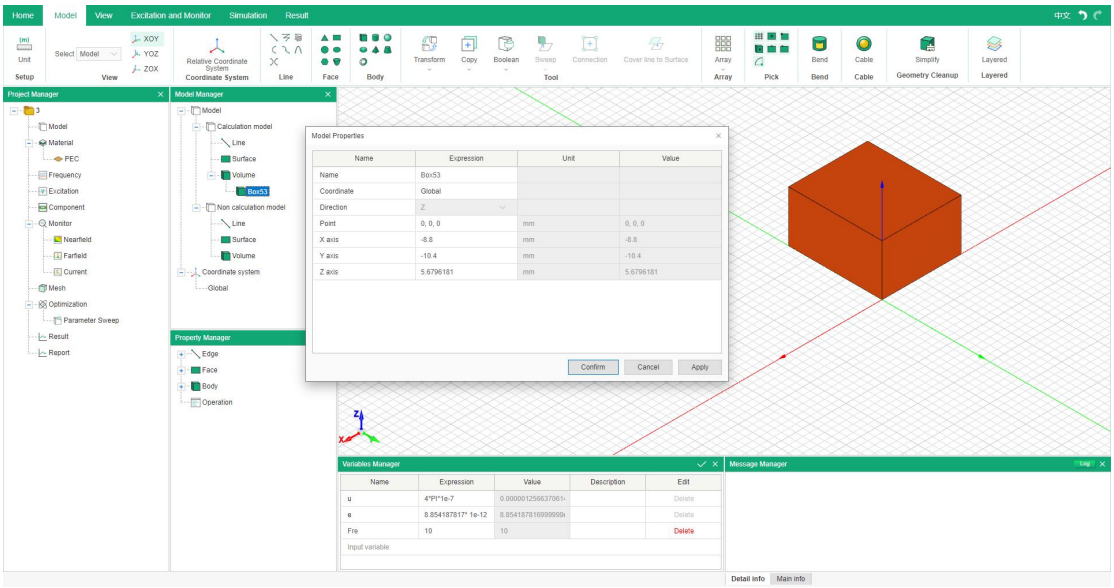


Figure 28 Box

2.6.2 Cylinder

The steps to create a cylinder model are:

1、 In the **Body** model column of the **Model** section, click the **Cylinder** button.

2、 Double-click the left mouse button in the viewing area to determine the center position of the bottom surface of the cylinder.

3、 Then move the mouse and click the left button of the mouse to

determine the radius of the bottom surface of the cylinder.

4、 Then move the mouse, double- click the left mouse button to determine the height of the cylinder.

5、 In the pop-up window, enter the model label name, center, radius and height.

6、 Finally click **Confirm** to finish.

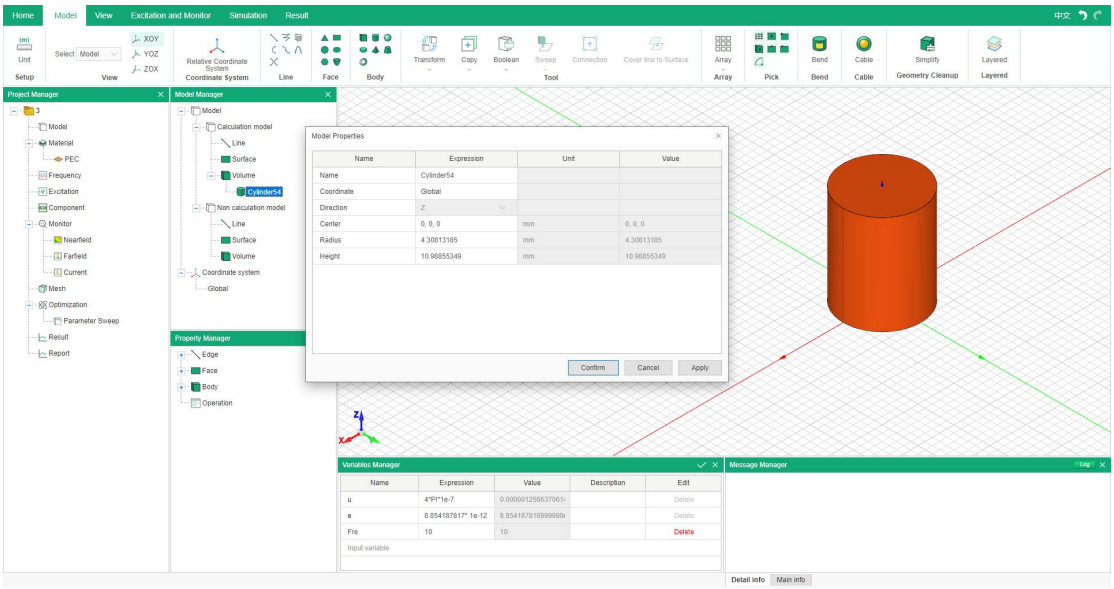


Figure 29 Cylinder

2.6.3 Sphere

The steps to build a ball model are:

1、 In the **Body** model column of the **Model** section, click the **Sphere** button.

2、 Double-click the left mouse button in the viewing area to determine the position of the center of the ball.

3、 Then move the mouse, double-click the left mouse button to determine the radius of the ball.

4、 In the pop-up pop-up window, enter the model label name, the position of the center of the sphere, and the radius.

5、 Finally click **Confirm** to finish.

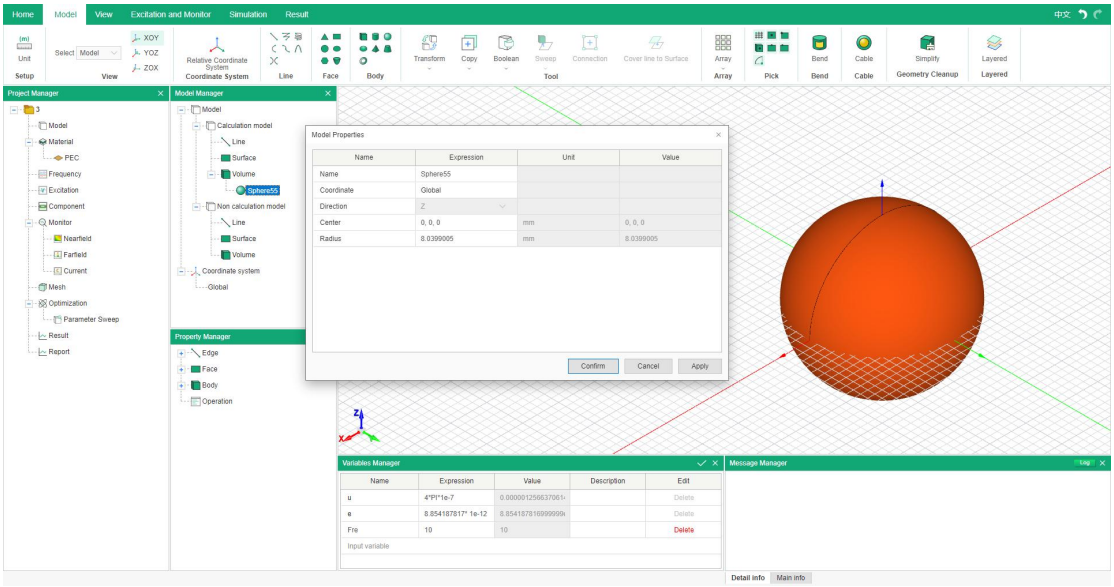


Figure 30 Sphere

2.6.4 Ellipsoid

The steps to create an ellipsoid model are:

1、 In the **Body** model column of the **Model** section, click the **Ellipsoid** button.

2、 Double-click the left mouse button in the viewing area to determine the position of the center of the ellipsoid.

3、 Move the mouse and click the left mouse button to determine the X-axis radius of the ellipsoid.

4、 Move the mouse and click the left mouse button to determine the Y-axis radius of the ellipsoid.

5、 Then move the mouse, double- click the left mouse button to determine the Z-axis radius of the ellipsoid.

6、 In the pop-up window, enter the model label name, center, and X/Y/Z axis radius.

7、 Finally click **Confirm** to finish.

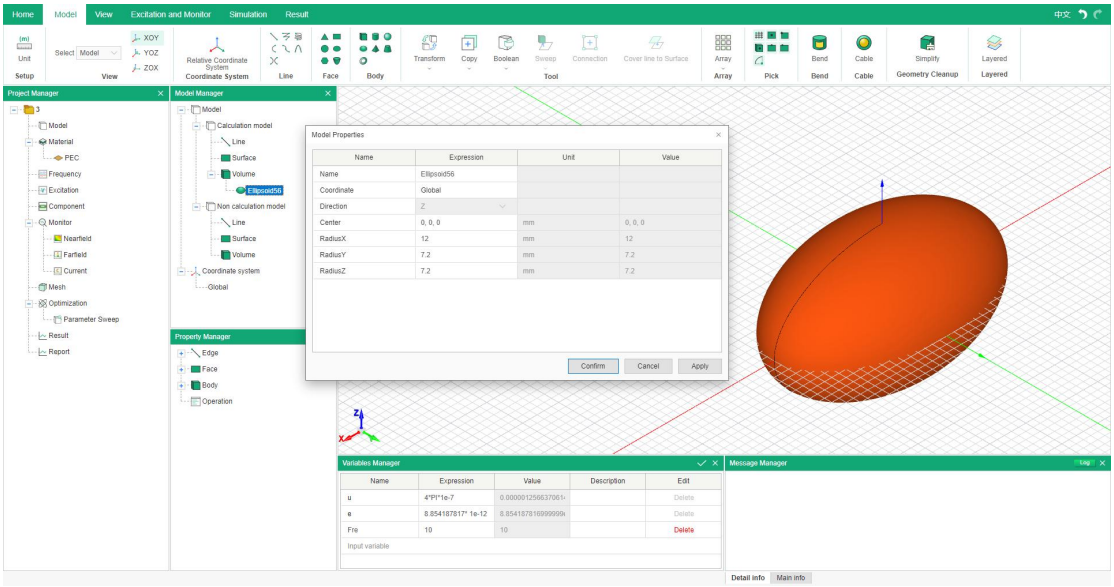


Figure 31 Ellipsoid

2.6.5 Four-Sided Platform

The steps to create a quadrangular truss model are:

1、 In the **Body** model column of the **Model** section, click the **Four-Sided Platform** button.

2、 Double-click the left mouse button in the viewing area to determine the apex position of the bottom surface of the quadrangular pedestal.

3、 Move the mouse and click the left button of the mouse to determine the size of the bottom surface of the four-sided pedestal.

4、 Move the mouse and click the left button of the mouse to determine the height of the quadrangular pedestal.

5、 Click the left button of the mouse in the viewing area to determine the position of the apex of the top surface of the quadrangular prism.

6、 Then move the mouse and click the left button of the mouse to determine the size of the top surface of the quadrangular pedestal.

7、 In the pop-up window, enter the model label name, vertex, height, base length, base width, top length, and top width.

8、 Finally, click **Confirm** to finish.

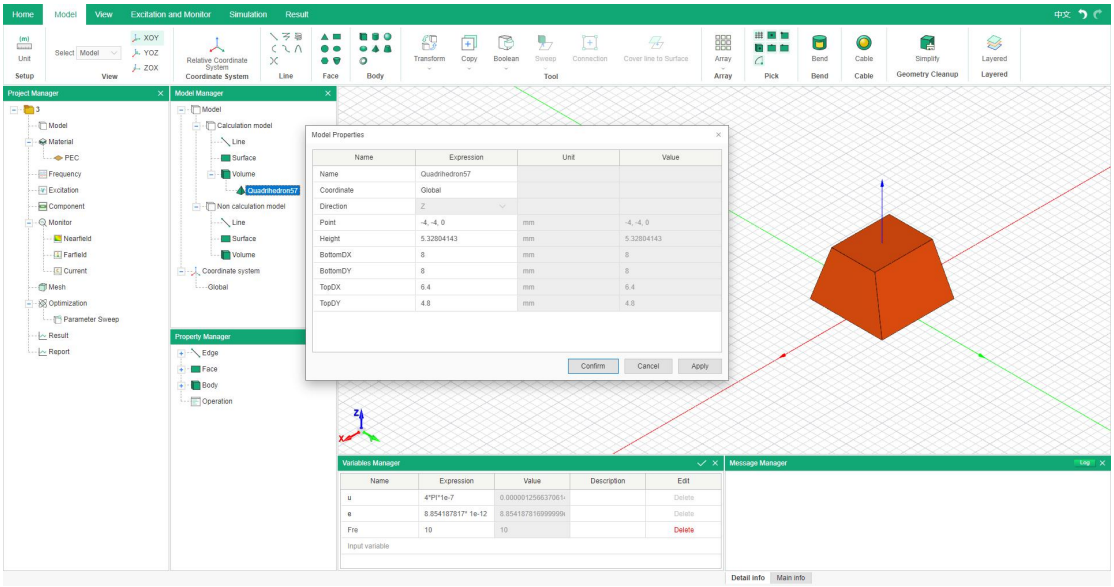


Figure 32 Four-sided platform

2.6.6 Cone

The steps to create a cone model are:

1、 In the **Body** model column of the **Model** section, click the **Cone** button.

2、 Double-click the left mouse button in the viewing area to determine the center position of the bottom surface of the circular platform.

3、 Move the mouse and click the left button of the mouse to determine the radius of the bottom surface of the circular table.

4、 Move the mouse and click the left button of the mouse to determine the height of the round table.

5、 Move the mouse, double- click the left mouse button to determine the radius of the top surface of the circular table.

6、 In the pop-up window, enter the model label name, center, height, top radius, and bottom radius.

7、 Finally, click **Confirm** to finish.

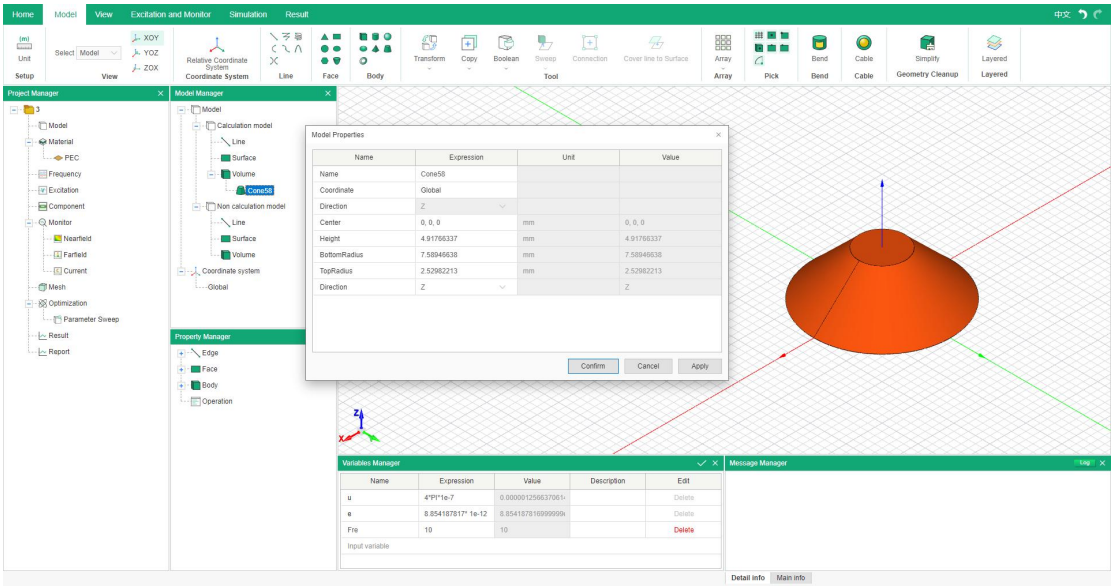


Figure 33 Cone

2.6.7 Torus

The steps to create a torus model are:

1、 In the **Body** model column of the **Model** section, click the **Torus** button.

2、 Double-click the left mouse button in the view area to determine the center position of the ring.

3、 Move the mouse and click the left button of the mouse to determine the radius of the inner circle of the ring.

4、 Move the mouse, double- click the left mouse button to determine the outer radius of the ring.

5、 In the pop-up window, enter the model label name, ring radius, section radius and ring angle.

6、 Finally, click **Confirm** to finish.

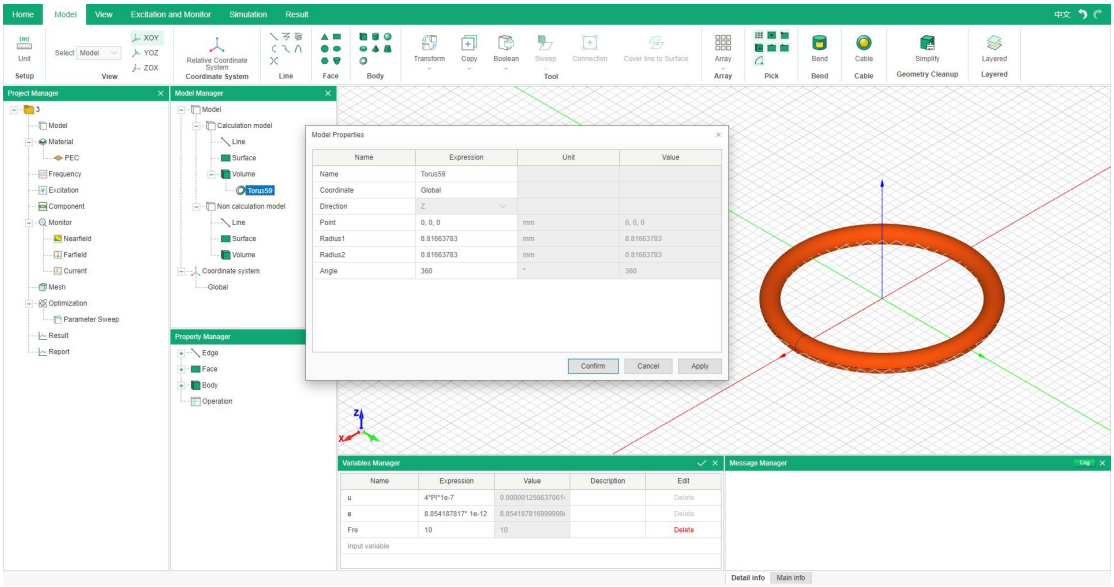


Figure 34 Torus

2.7 Tool

Tool includes **Transform**, **Copy**, **Boolean**, **Sweep**, **Connect** and **Cover line to surface**.

2.7.1 Transform

Transform includes **Translate**, **Rotate**, **Scale**, and **Mirror**.

Button	Button Label Name	Function
	Translate	Translate the model model
	Rotate	Rotate the model
	Scale	Scale the model
	Mirror	Mirror the model

2.7.1.1 Translate

The specific steps of the **Transform-Translate** operation are:

1、 Select the tag name of the model to be translated in the model manager area.

2、 In the toolbar of the modeling section, click the **Transform -Translate** button.

3、 Move the mouse to the viewing area, and double-click the mouse to determine the starting point of translation.

4、 Continue to move the mouse, double-click the mouse to determine the end point of translation.

5、 In the pop-up box, enter the translation vector value and click **Confirm** to finish.

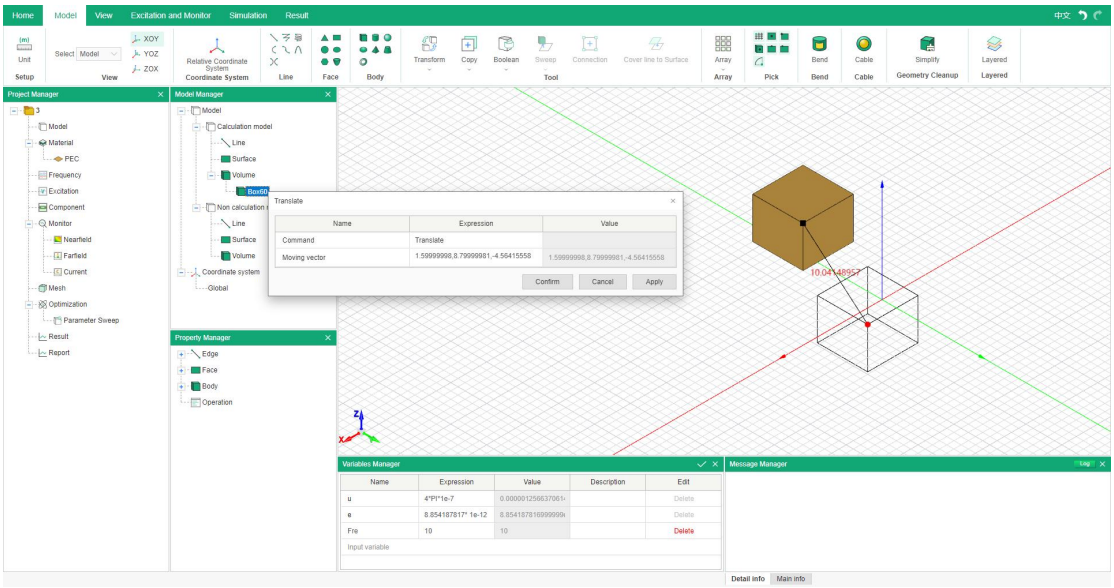


Figure 35 Transform-translate

2.7.1.2 Rotate

The specific steps of the **Transform- Rotate** operation are:

1、 Select the tag name of the model to be rotated in the model manager area.

2、 In the toolbar of the modeling section, click the **Transform -Rotate** button.

3、 In the pop-up box, select the rotation axis, enter the rotation angle,

and click **Confirm** to finish.

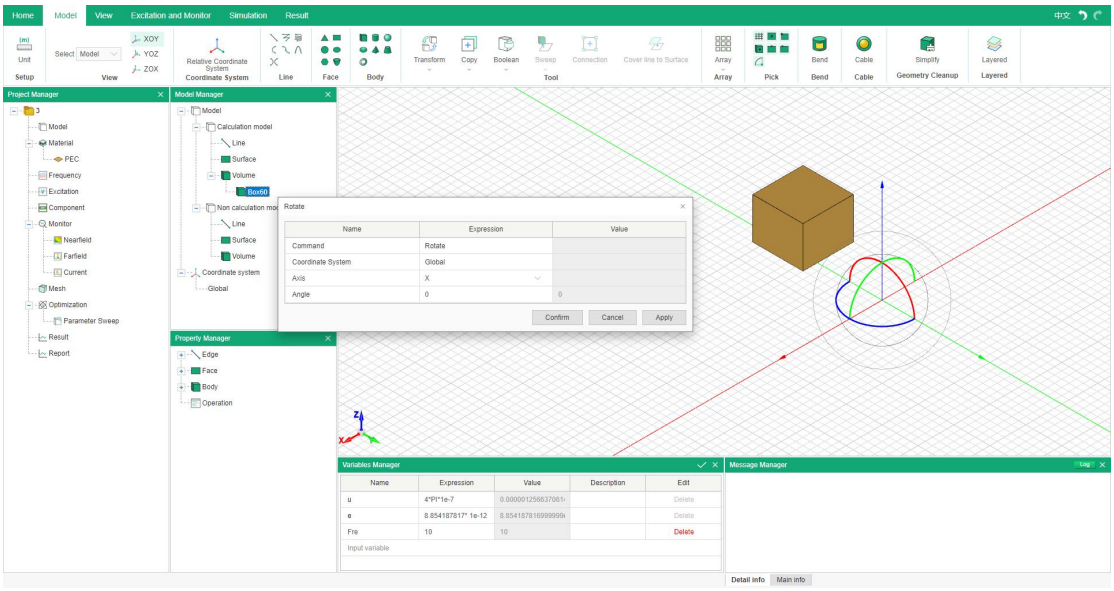


Figure 36 Transform- rotate

2.7.1.3 Scale

The specific steps of the **Transform- Scale** operation are:

1、 Select the tag name of the model to be scaled in the model manager area.

2、 In the toolbar of the modeling section, click the **Transform -Scale** button.

3、 In the pop-up box, enter the zoom size and click **Confirm** to finish.

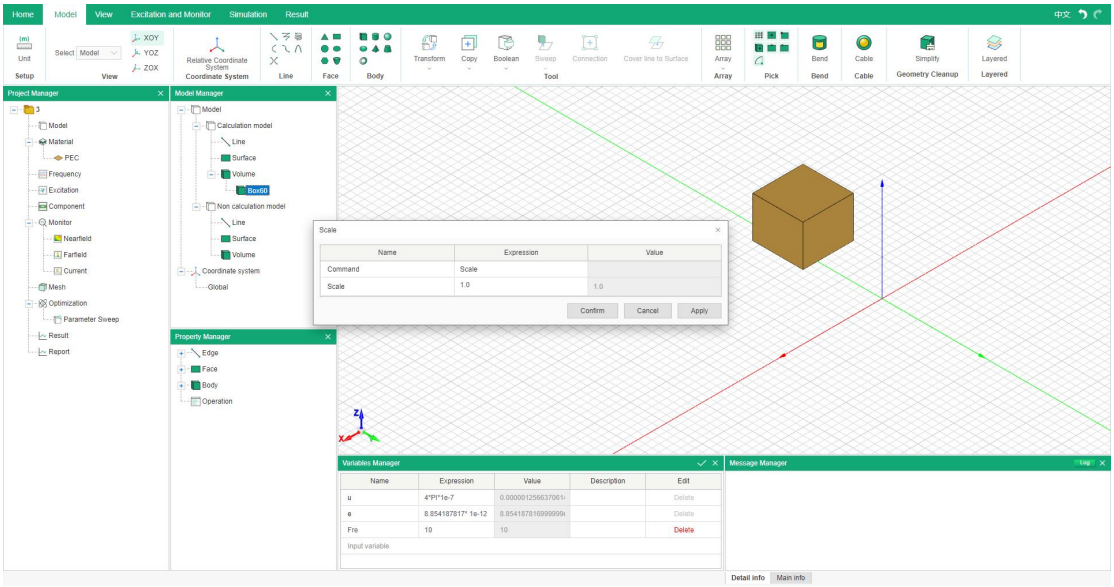


Figure 37 Transform- scale

2.7.1.4 Mirror

The specific steps of **Transform- Mirror** operation are:

1、 Select the tag name of the model to be mirrored in the model manager area.

2、 In the toolbar of the modeling section, click the **Transform -Mirror** button.

3、 In the pop-up box, enter the mirror symmetry point and the mirror symmetry normal vector, and click **Confirm** to finish.

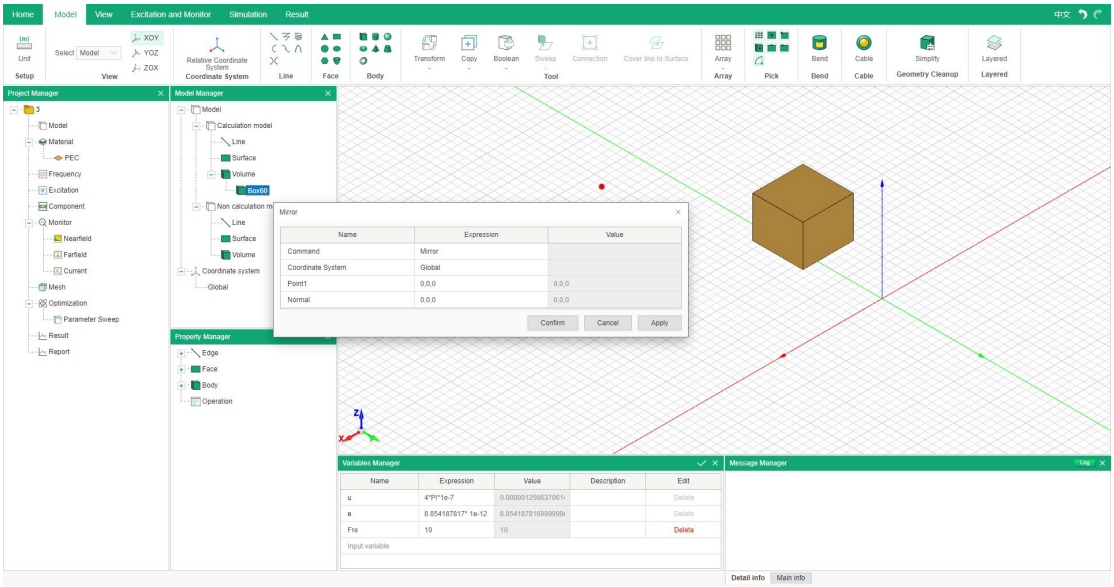


Figure 38 Transform- mirror

2.7.2 Copy

Copy includes **Translate**, **Rotate**, and **Mirror**.

Button	Button Label Name	Function
	Translate	Translate the models with the same distance between each model
	Rotate	Rotate the model with the same angle between each model
	Mirror	Make mirror symmetry to the model, mirror symmetry between each model

2.7.2.1 Translate

The specific steps of the **Copy -Translate** operation are:

1、 In the model manager area, select a model's tag.

2、 In the toolbar of the modeling section, click the **Copy -Translate**

button.

3、 Move the mouse to the viewing area, and double-click to determine the starting point of translation.

4、 Continue to move the mouse, double-click to determine the end point of translation.

5、 In the pop-up box, enter the translation vector value and click **Confirm** to finish.

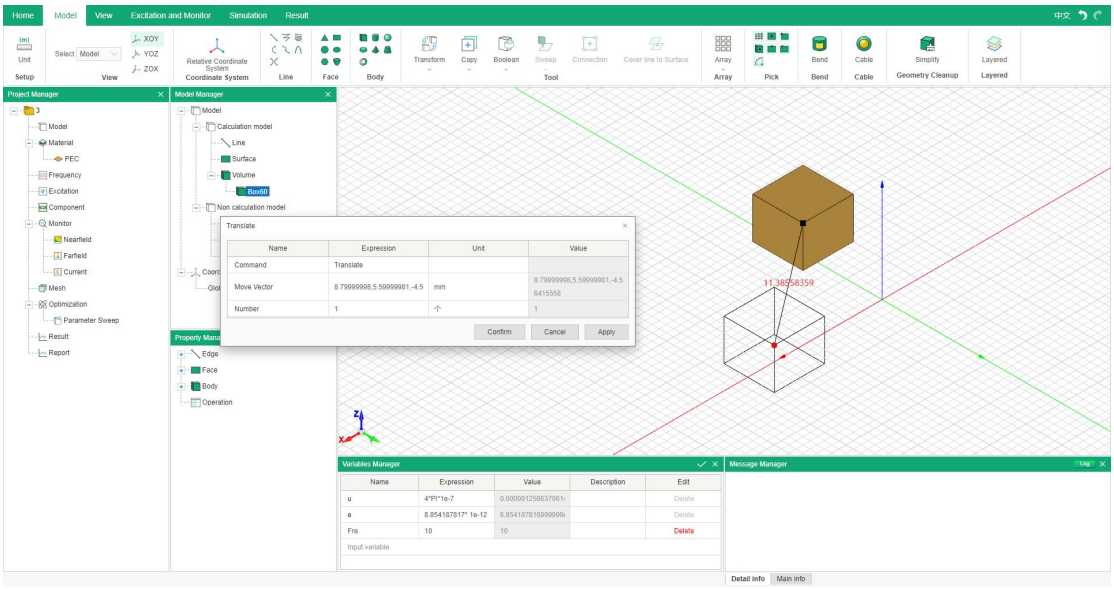


Figure 39 Copy-translate

2.7.2.2 Rotate

The specific steps of the **Copy - Rotation** operation are:

1、 In the model manager area, select a model's tag.

2、 In the toolbar of the modeling section, click the **Copy-Rotate** button.

3、 In the pop-up box, select the rotation axis, enter rotation angle, and click **Confirm** to finish.

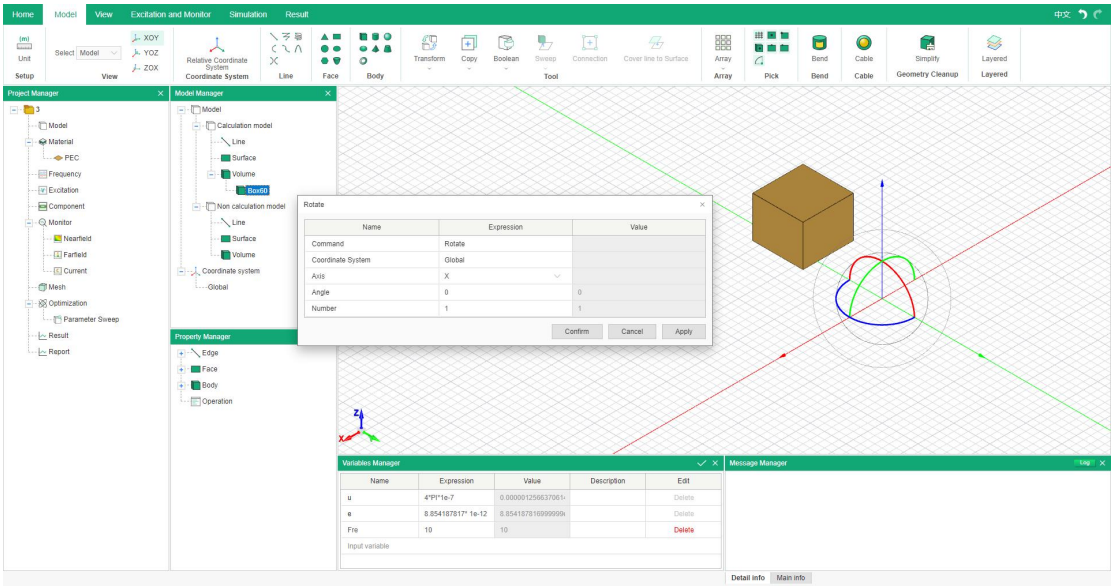


Figure 40 Copy-rotate

2.7.2.3 Mirror

The specific steps of **Copy - Mirror** symmetry operation are:

1、 In the model manager area, select a model's tag.

2、 In the toolbar of the modeling section, click the **Copy-Mirror** button.

3、 In the pop-up box, enter the mirror symmetry point and the mirror symmetry normal vector, and click **Confirm** to finish.

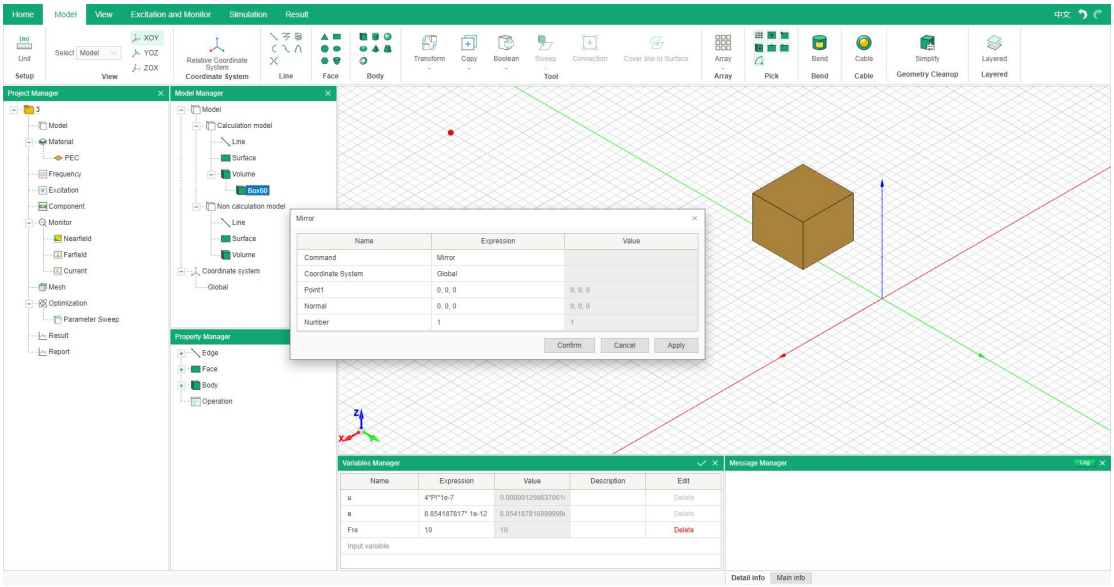


Figure 41 Copy-mirror

2.7.3 Boolean

Boolean includes **Union**, **Add**, **Intersect**,**Subtract**, and **Split.**

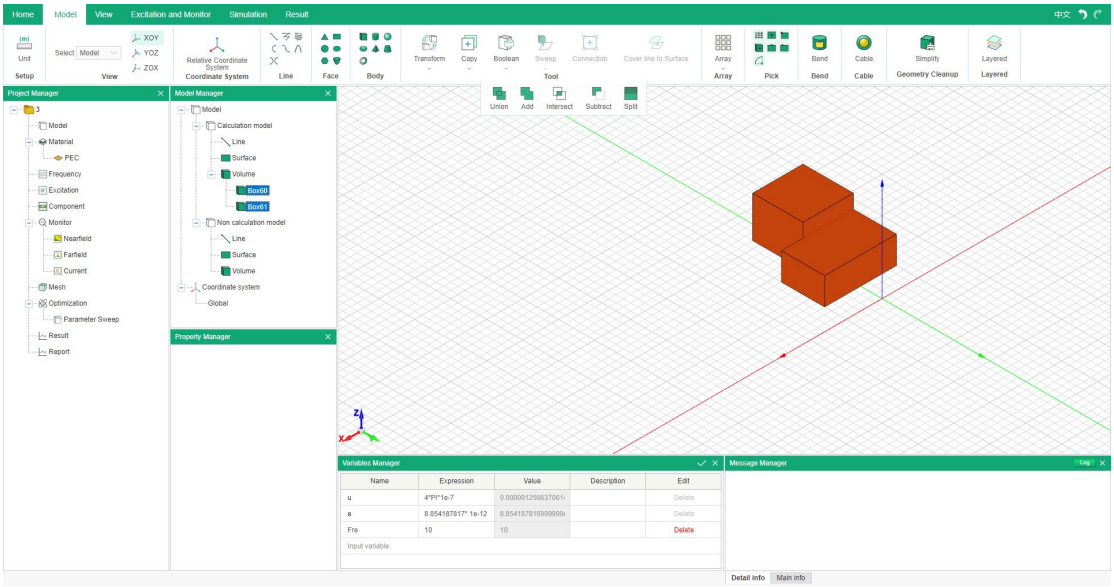
Button	Button Label Name	Function
	Add	Add the model
	Intersect	Intersect the model
	Union	Combine the models
	Subtract	Subtract the models
	Split	Separate operations on the model

2.7.3.1 Add

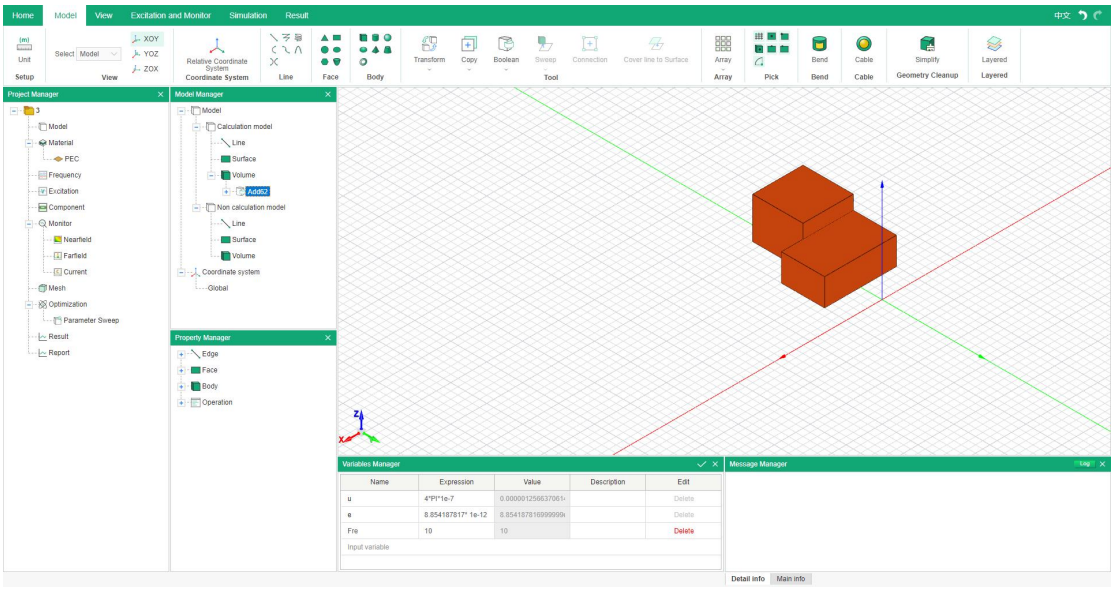
The specific steps of the **Boolean-Add** operation are:

1、 In the model manager area, select a model's tag and hold down **Ctrl** to perform multiple selection.

2、 Click **Boolean-Add** button.



(a) Boolean-add



(b) After boolean-add

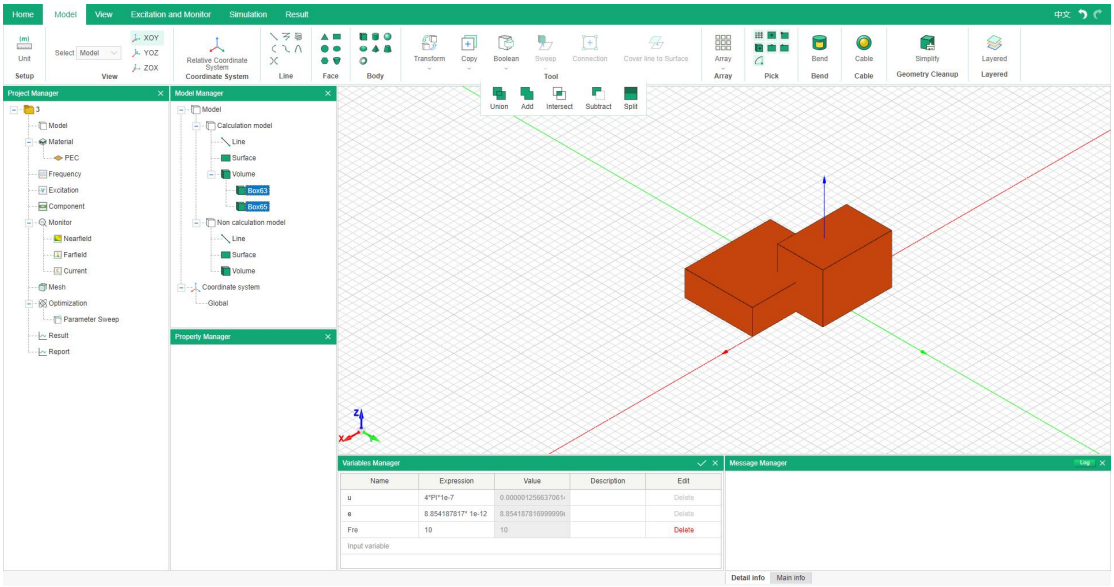
Figure 42 Boolean-add

2.7.3.2 Intersect

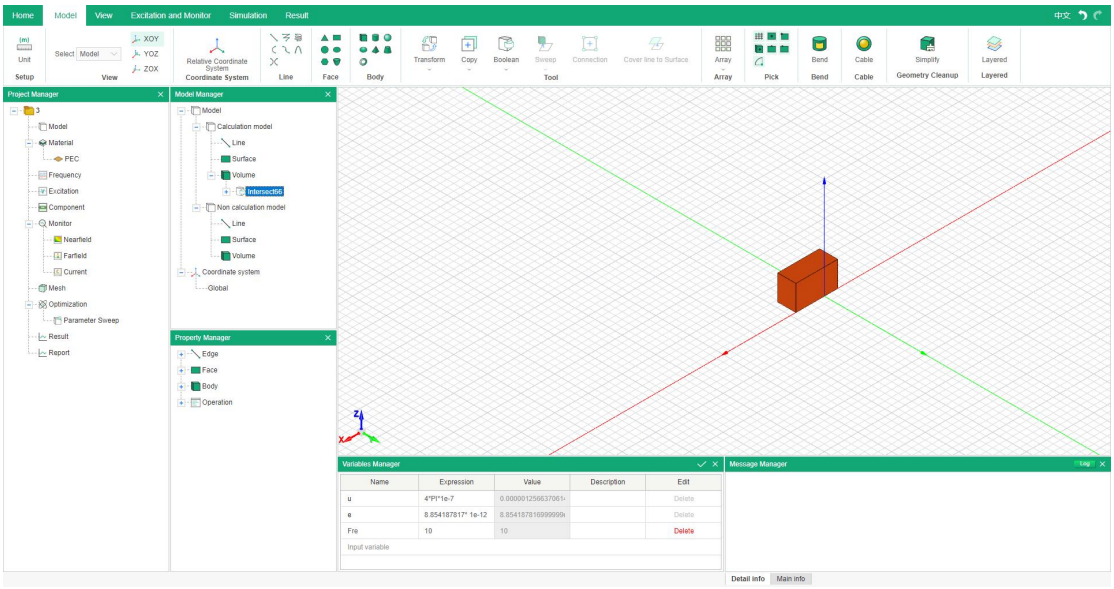
The specific steps of **Boolean-Intersect** operation are:

1、 In the model manager area, select a model's tag and hold down **Ctrl** to perform multiple selection.

2、 Click **Boolean-Intersect** button.



(a) Boolean-intersect



(b) Boolean-after intersection

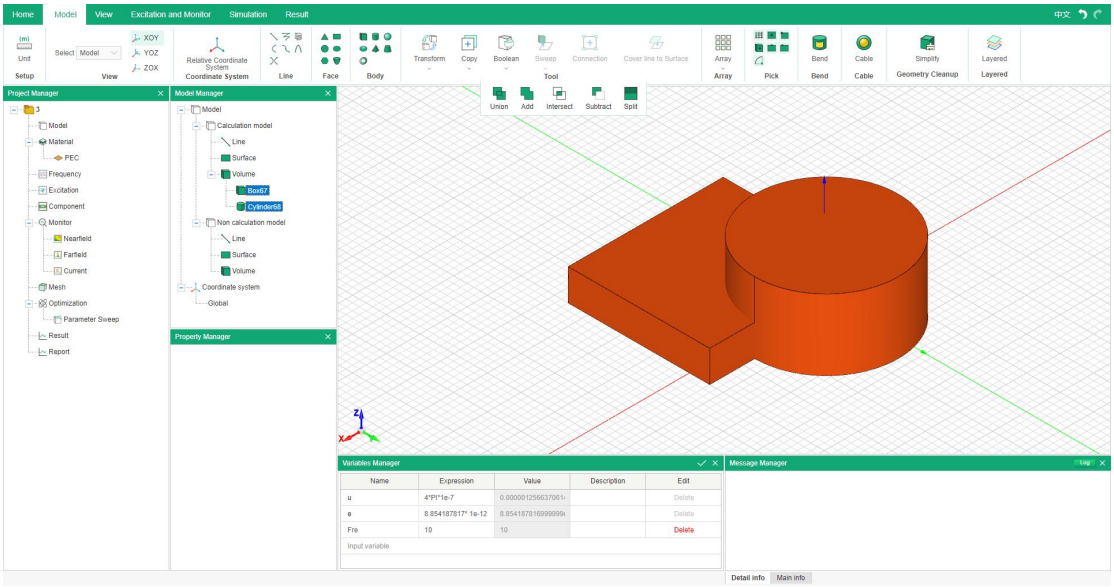
Figure 43 Boolean-intersect

2.7.3.3 Union

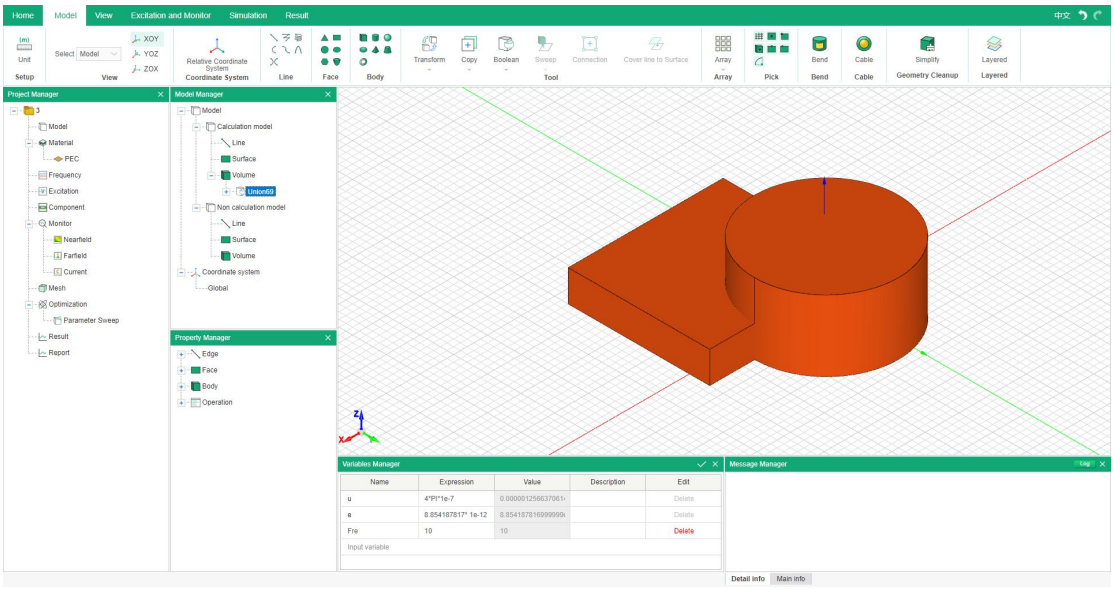
The specific steps of **Boolean-Union** operation are:

1、 In the model manager area, select a model's tag and hold down **Ctrl** to perform multiple selection.

2、 Click **Boolean-Intersect** button.



(a) Boolean-union



(b) Boolean-after union

Figure 44 Boolean-Union

2.7.3.4 Subtract

The specific steps of **Boolean-Subtract** operation are:

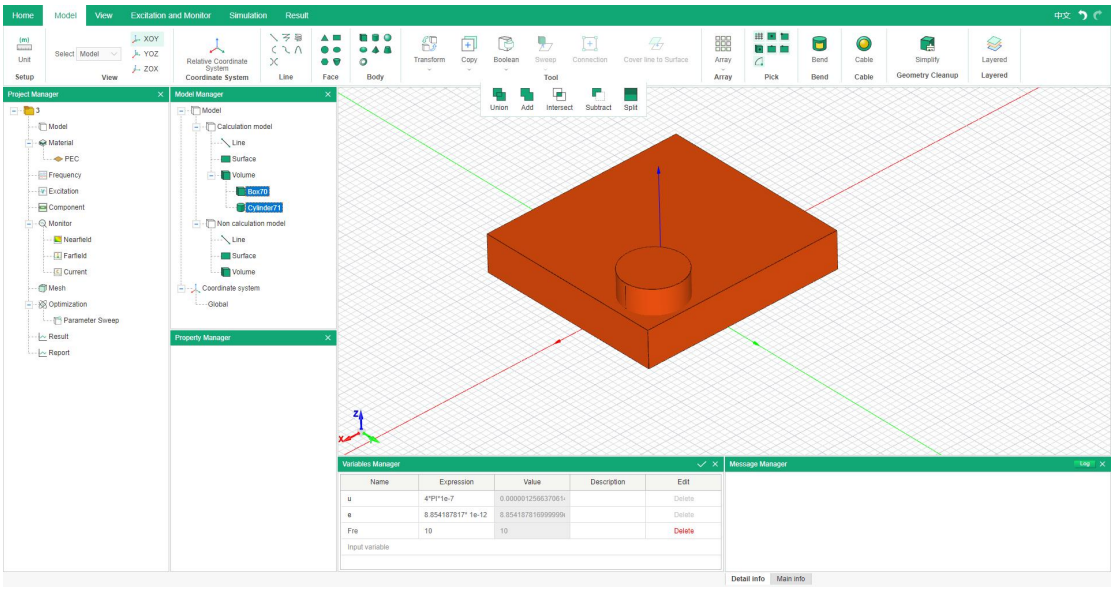
1、 In the model manager area, select a model's tag and hold down **Ctrl** to perform multiple selection.

2、 Under Tools in the section, click the Boolean Operation - Subtract button.

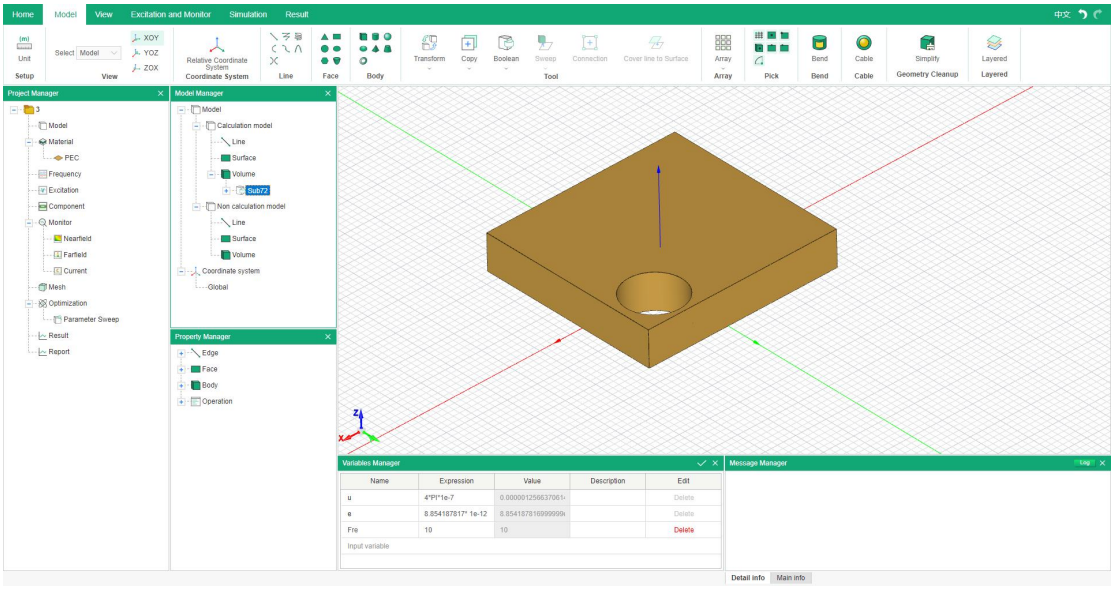
3、 In the pop-up drop-down box, click to select the subtraction label name.

4、 In the pop-up window, select the subtracted model and the removed model, and check whether to keep the removed model.

5、 Click **Confirm**.



(a) Boolean-subtract



(b) Boolean-after subtract

Figure 45 Boolean -subtract

2.7.4 Sweep

Sweep includes **Sweep Along Vector**, **Sweep Along Path**, and **Sweep Along Axis**.

Button	Button Label Name	Function
	Sweep Along Vector	Sweep the model along the vector
	Sweep Along Path	Sweep the model along a path
	Sweep Along Axis	Sweep the model along the coordinate system

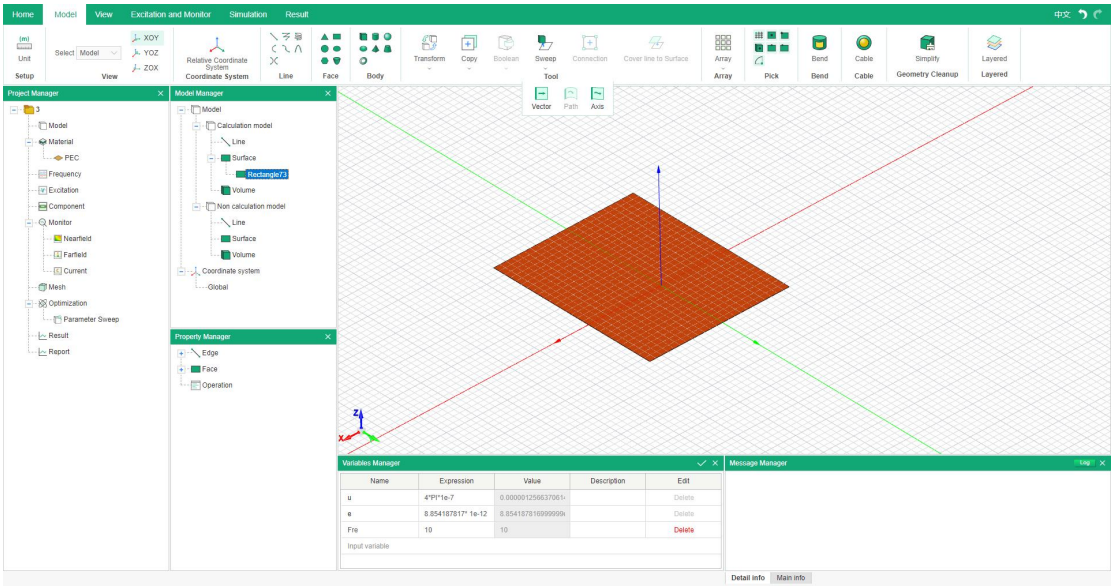
The **Sweep** operation is illustrated by **Sweep Along Vector**.

1、 Select a surface model in the view area or select the label of a certain surface model in the model manager area.

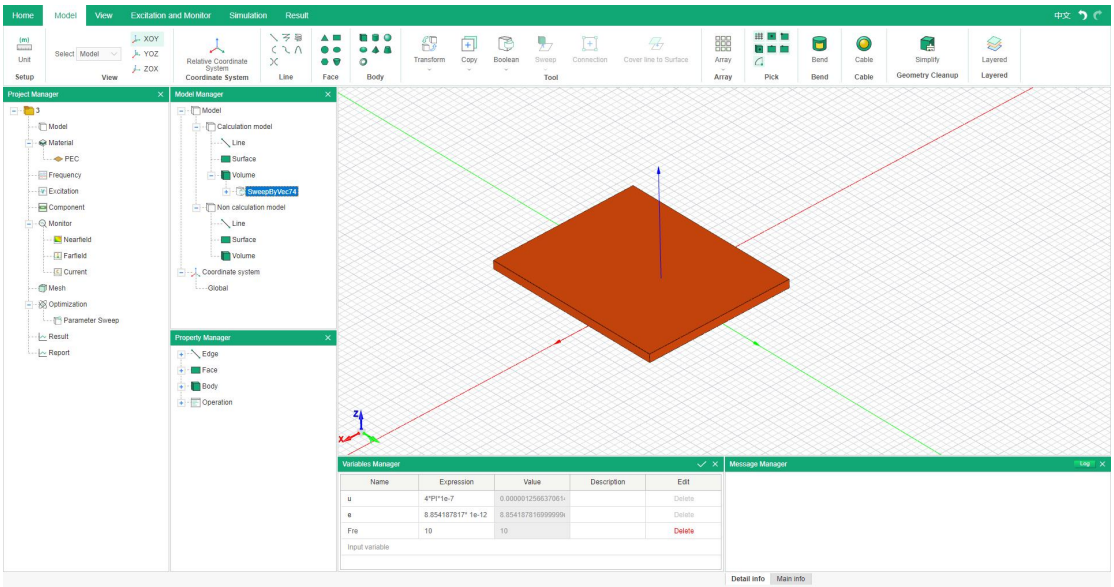
2、 Click the **Sweep** tab in the red box of the section, and then click the **Vector** tab.

3、 In view area, double-click the right mouse button, select a point 1, then double-click the right mouse button to complete the selection of point 2. The box behind the vector of the pop-up window along the vector sweep will display P2-P1 value (users can also modify this parameter manually).

4、 Click **Confirm.**



(a) Sweep - sweep along vector



(b) Sweep - after sweeping along vector

Figure 46 Sweep - Sweep along vector

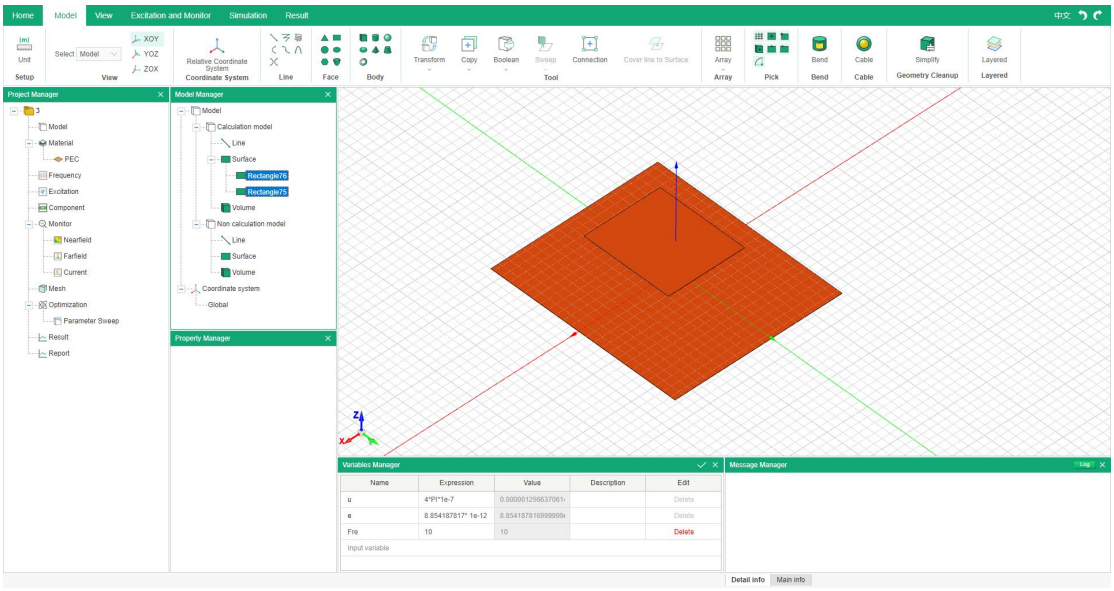
2.7.5 Connect

Connect couples two planes to generate a solid model.

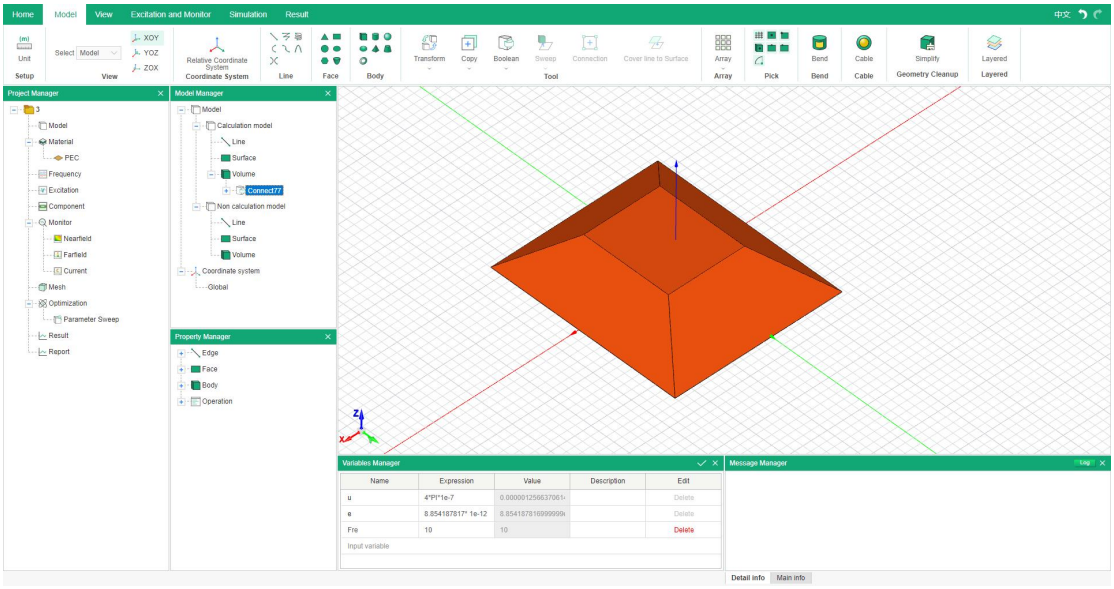
The steps of connect operation are:

1、 Hold down **Ctrl** in the view area, and select two surface models (or hold down the **Ctrl** in the model manager area, and select the labels of two surface models).

2、 Click **Connect**.



(a) Connect



(b) After connect

Figure 47 connect

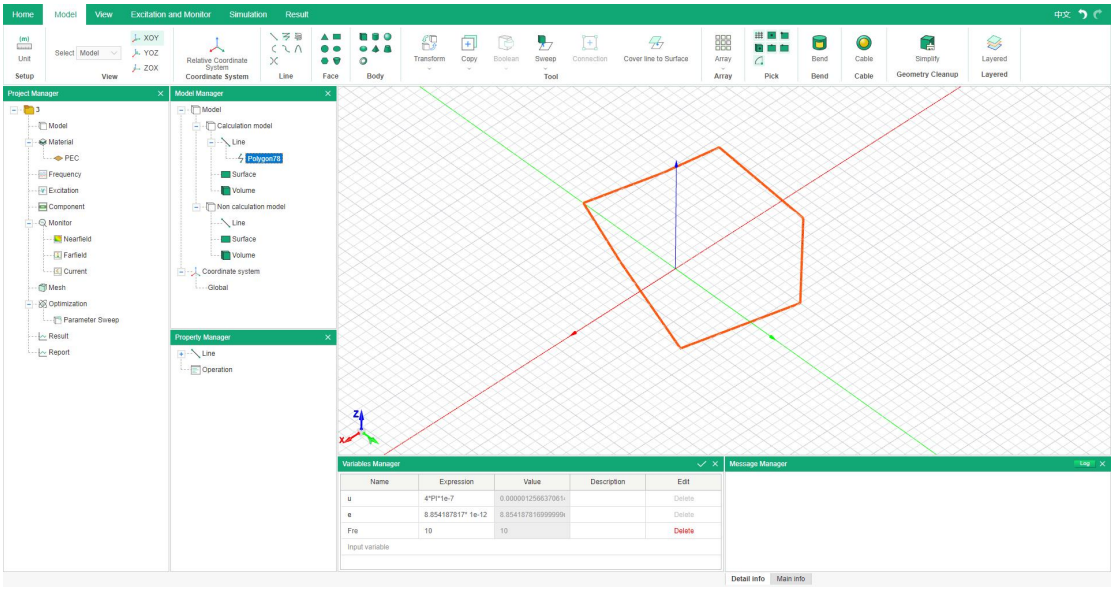
2.7.6 Cover Line To Surface

Cover Line To Surface generates a surface from a closed multi-line segment model.

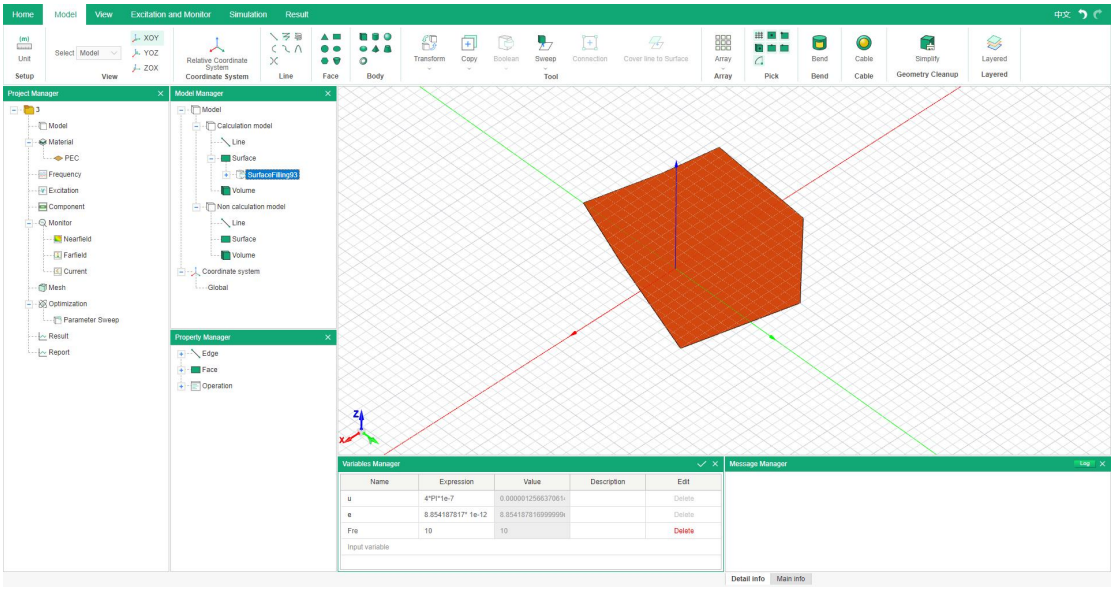
The steps of cover line to surface operation are:

1、 In the view area, select a closed multi-line segment model or select the label name of a closed multi-line segment model in the model manager area.

2、 Click **Cover Line To Surface**.



(a) Cover line to surface



(b) After cover line to surface

Figure 48 Cover line to surface

2.8 Array

Array generation function quickly generates a visual array layout with specific

array spacing and array element number.

Array includes **Linear Array** and **Rectangular Array**.

Take the line array as an example to illustrate the operation of the array.

1、 In the model manager area, select the volume model to be generated (select the label name of the model).

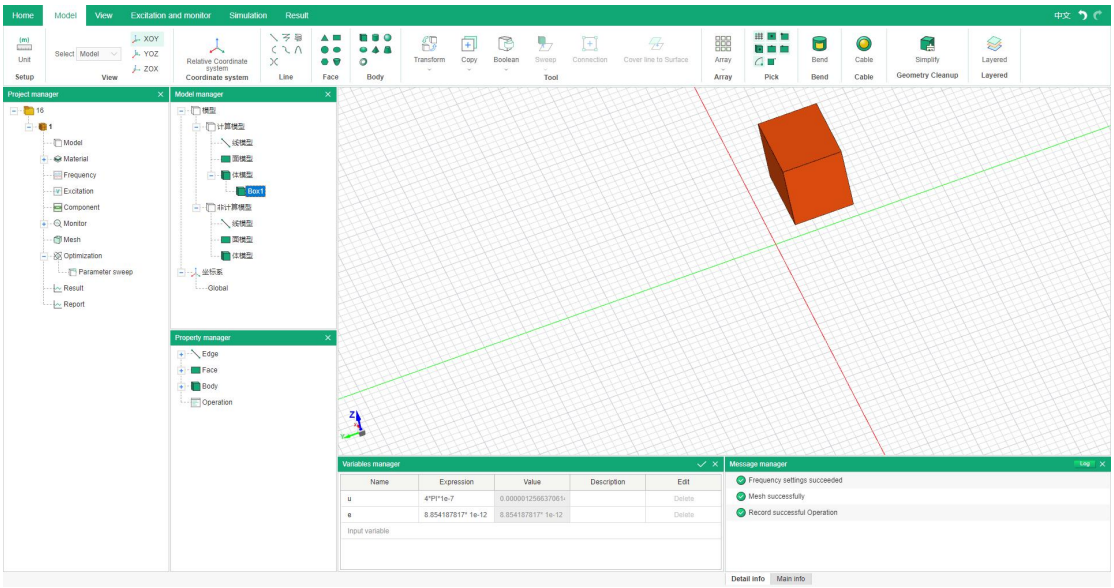
2、 Under the modeling area, click **Array**.

3、 Click **Line Array** in the pop-up drop-down box.

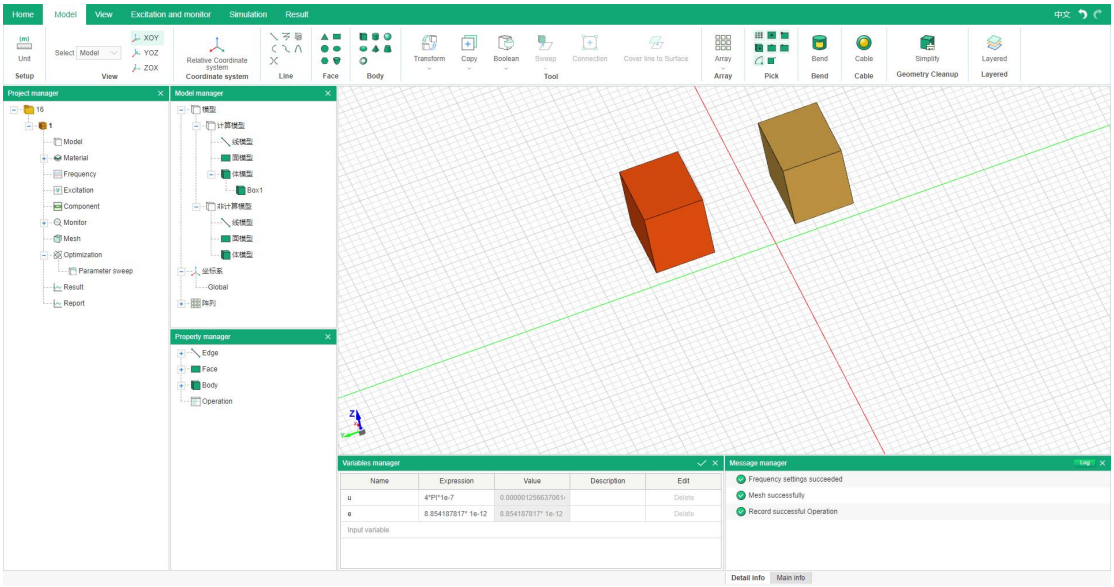
4、 In the pop-up pop-up window, select the coordinate axis (x-axis)

5、 Enter the number and interval.

6、 Click **Confirm**.



(a) Array generation



(b) After generating the array

Figure 49 Array Generation

2.9 Pick

Pick quickly selects points on the established model as the vertices of the model during interactive modeling.

Pick includes **Grid Point, Model Vertex, Midpoint Of Edge, 1/4 Point Of Edge, Midpoint Of Surface, Center Of Body, Center Of Arc**.

Button	Button Label Name	Function
	Grid Point	Quickly select grid points
	Model Vertices	Quickly select model vertices
	Midpoint Of Edge,	Quickly select the midpoint of an edge
	1/4 Point Of Edge	Quickly select 1/4 points of sides
	Midpoint Of Surface	Quickly select the midpoint of a face
	Center Of Body	Quick select the center of the body
	Center Of Arc	Quickly select the center of the

		arc

2.10 Bend

Bend maps a face model to a volume model surface.

The specific steps of the mapping operation are:

1、 Select a surface model and a volume model to be mapped to in the view area or select the label names corresponding to the surface model and volume model in the model manager area.

2、 On the section, click **Bend**.

3、 In the pop-up window, select the surface model as the attachment surface, and the volume model as the attached surface.

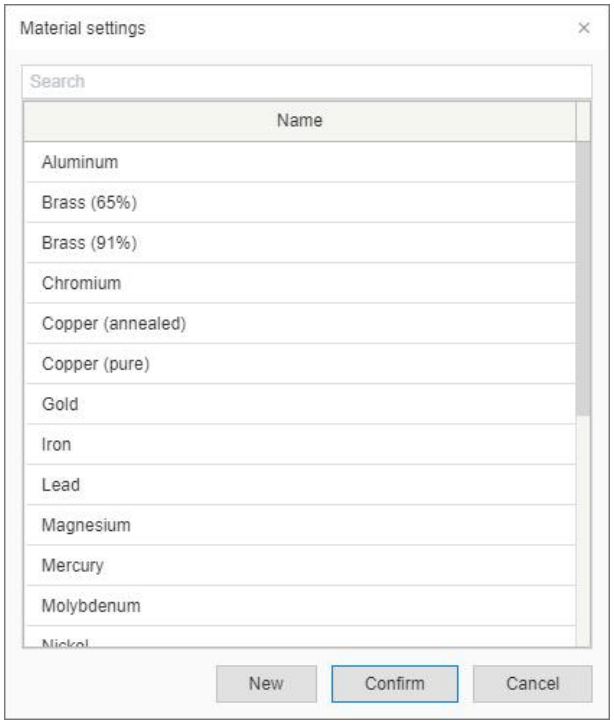
4、 Click **Comfirm**.

3 Material Setting Module

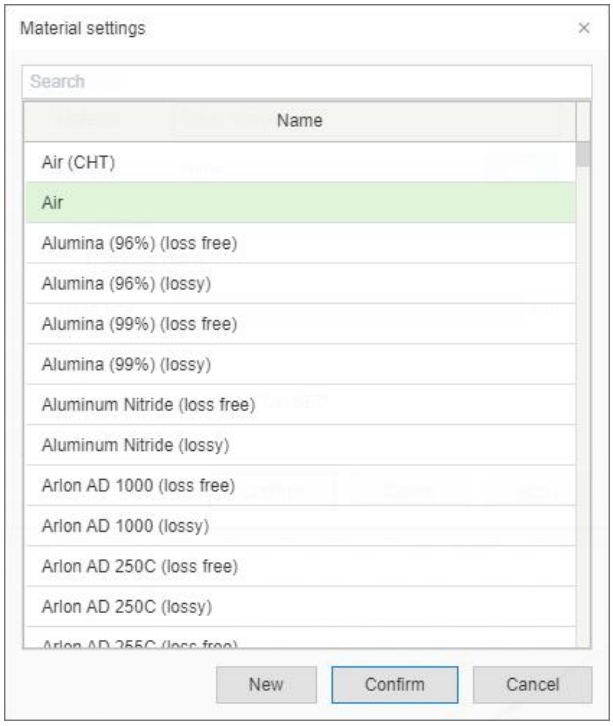
Material settings set up the model material. The material includes **dielectric** material, **metal** material and **PEC**. The surface model can only be setup as metal and PEC (the default material of the surface model is PEC), and the volume model can only be setup as dielectric material and PEC (the default material of the volume model is PEC).

3.1 Material library

RDSim has two material libraries: the dielectric material library and the metal material library, both of which include common material parameters. The metal material library is used for the material setting of the surface model and the surface on the solid model. The dielectric material library is used for the material setting of the solid model.



(a) Metal material library



(b) Dielectric material library

Figure 50 Material library

3.2 Material search

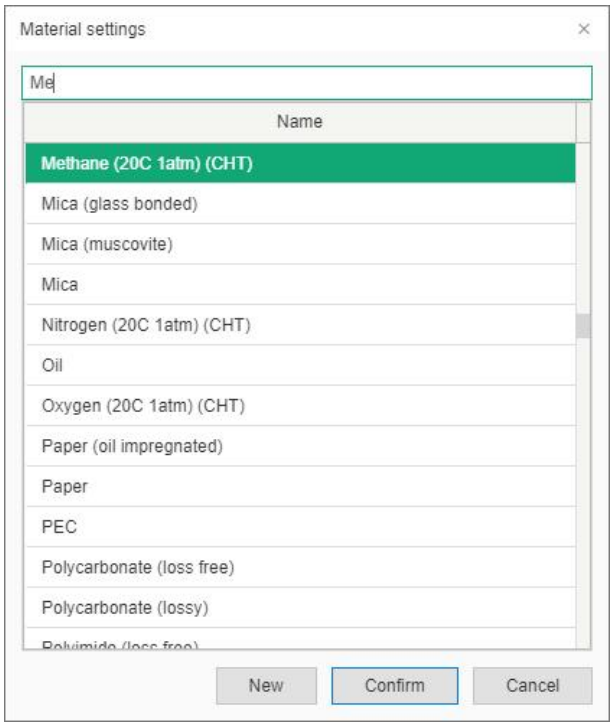


Figure 51 Material search

The material library supports the material search function, which is convenient for users to quickly set materials.

The operation steps of material search in material library are as follows:

1、 Enter some or all of the label names of the materials in the library search box.

2、 After entering the label name, the material will appear automatically.

3.3 New material

The new material option is for users to set up custom materials.

3.3.1 New dielectric material

1、 Click **New** in the material library or right-click the material button in the project manager, and click **New Material**.

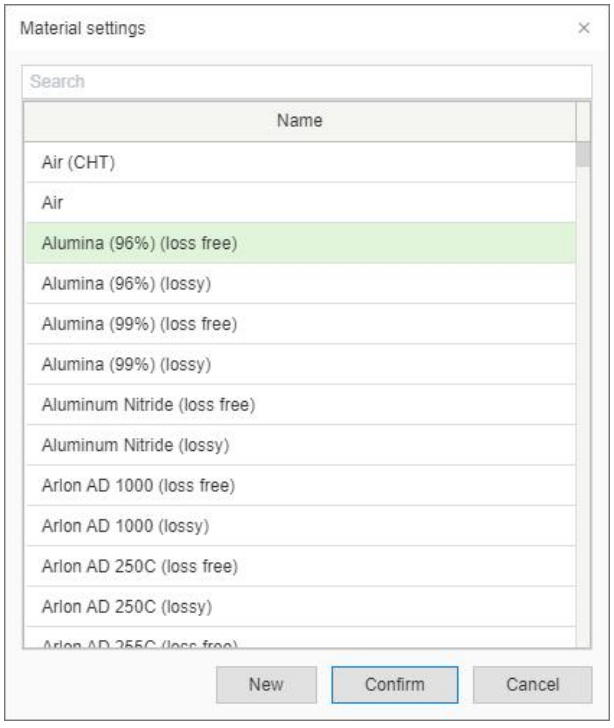


Figure 52 New material 1

2、 In the new material window, select the material type as dielectric, and enter the material label name, material color and transparency in turn.

Select whether to display the model wire frame and whether to display the material color in the selection (the default is display).

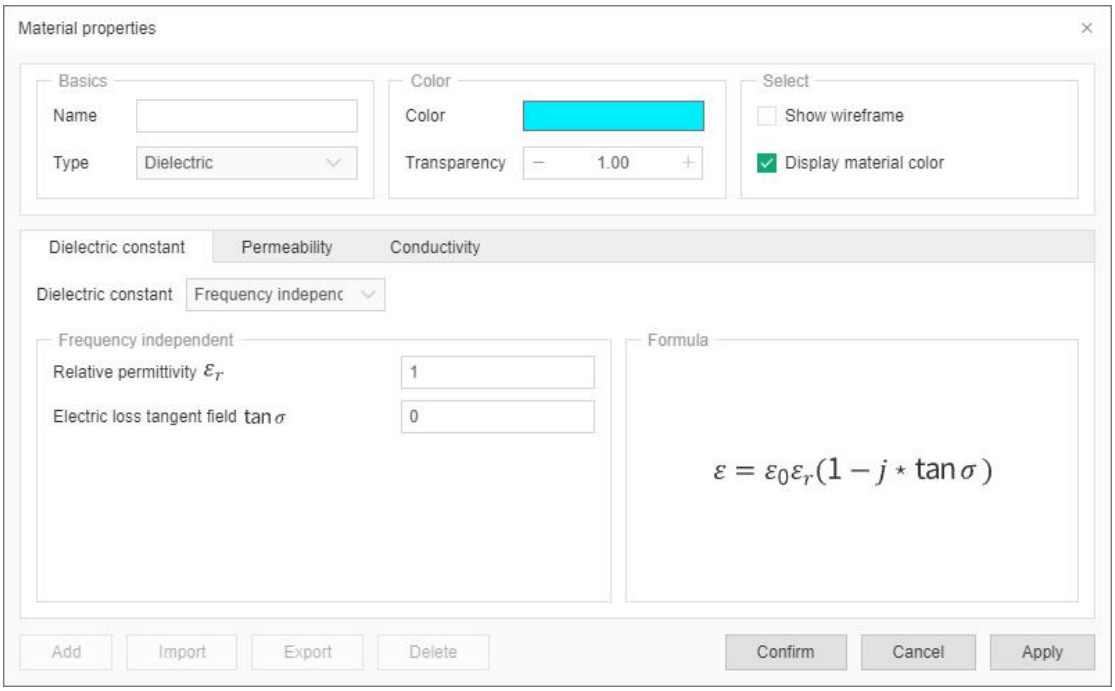


Figure 53 New material

3、 Under the dielectric constant, there are frequency independent, input list, Debye Relaxation, Cole-Cole, Havriliak-negami and Djordjevic-Sarkar options. In each option, the user can enter different parameters to set the dielectric constant.

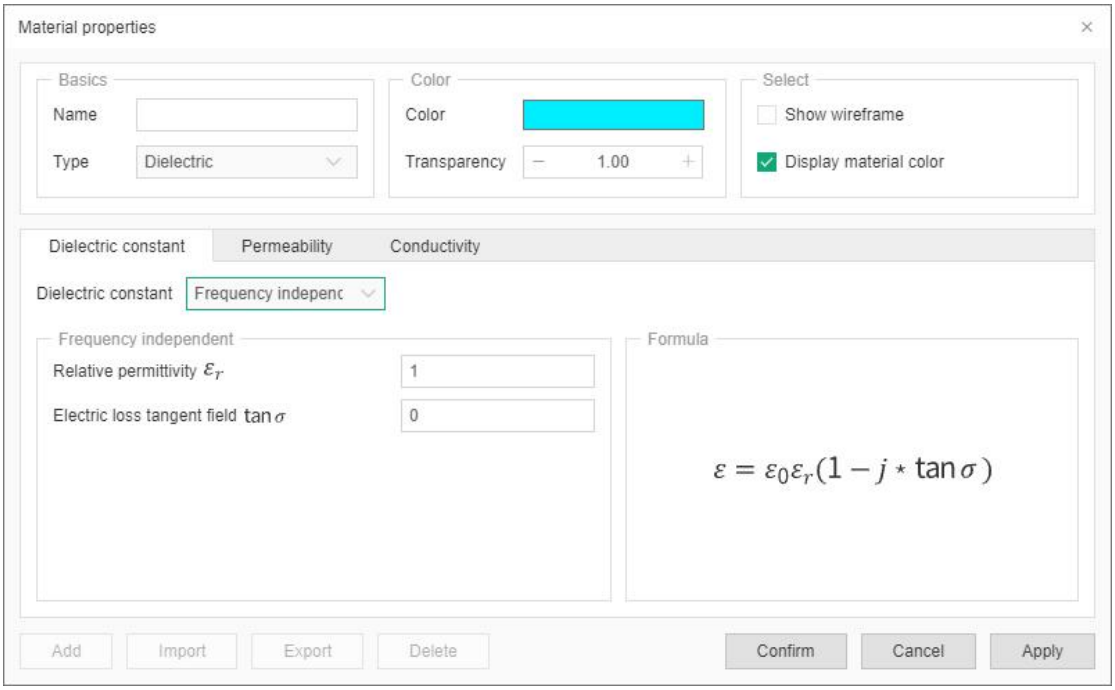


Figure 54 Dielectric constant

4、 Under magnetic permeability, there are two options: independent of frequency and input list. In each option, the user can input the parameters of magnetic permeability.

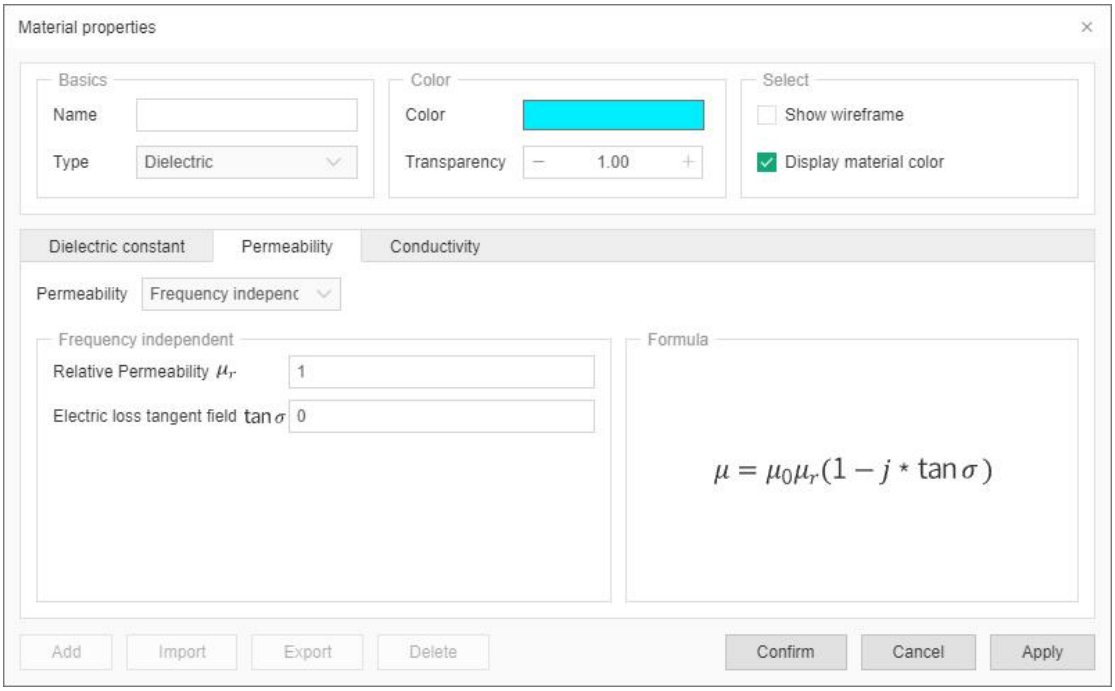


Figure 55 Permeability

5、 Under conductivity, the user can input the value of conductivity (if there is no conductivity, it can be left blank).

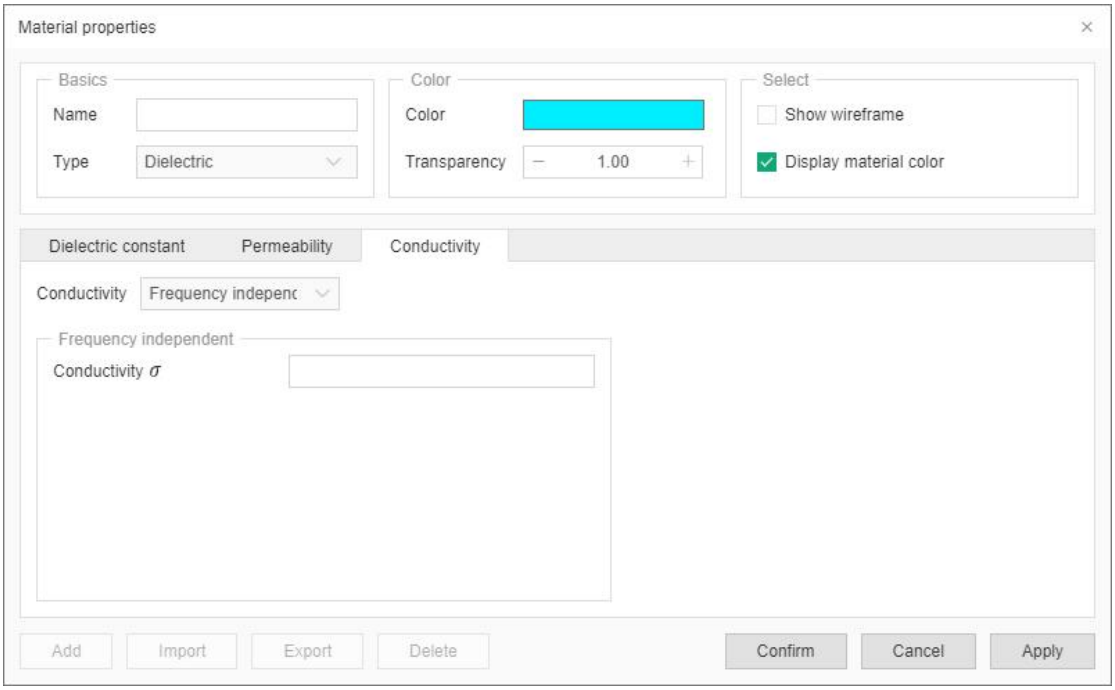


Figure 56 Conductivity

Note: If choose to enter the list, click the **Import** button to import the corresponding parameters from file.

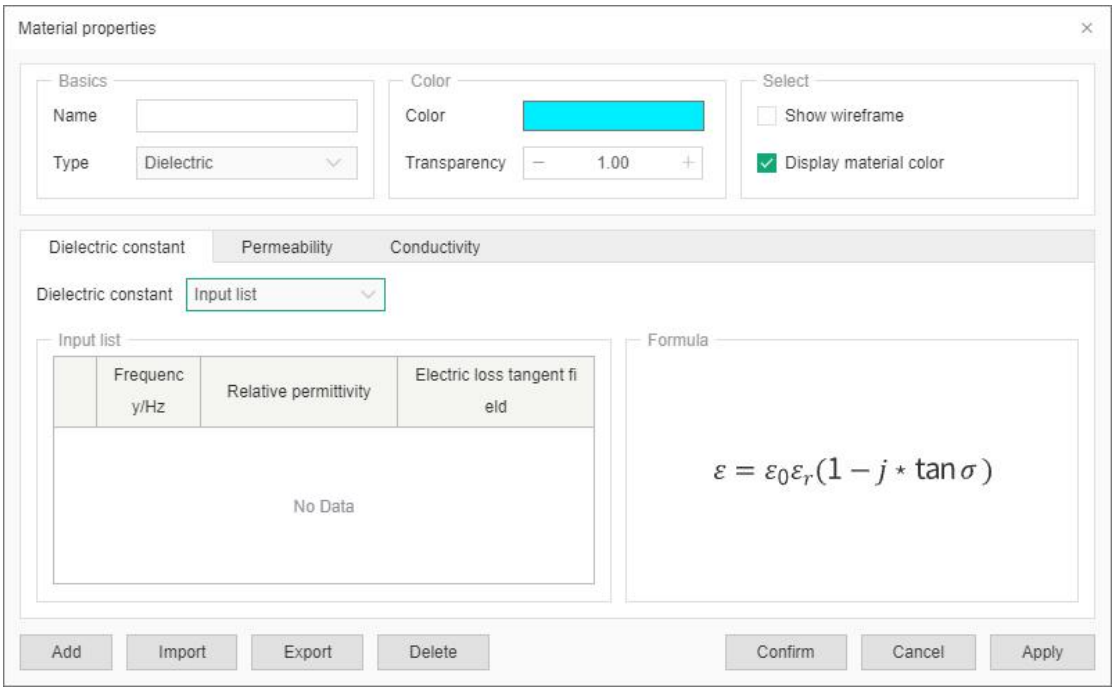


Figure 57 New Material - import

3.3.2 New metal material

1、 Click the **New** in the material library or right-click the material button in the project manager, and click **New Material.**

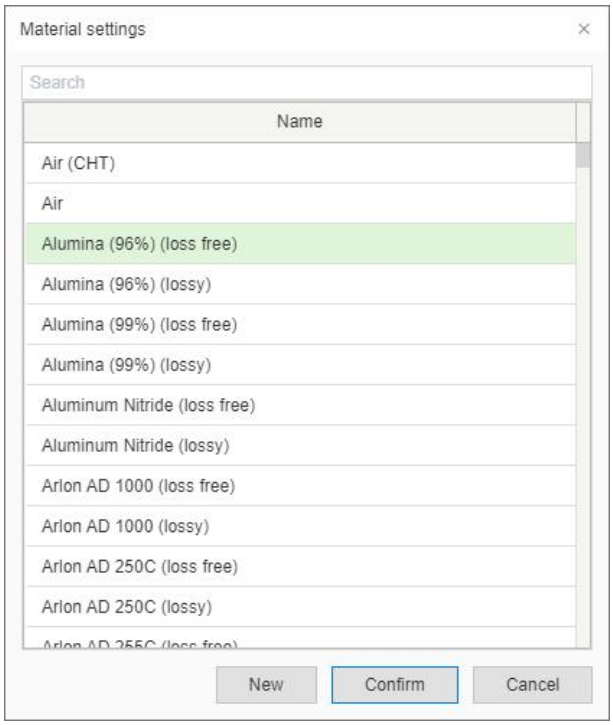


Figure 58 New material 1

2、 In the new material window, select the material type as metal, and enter the material label name, material color and transparency in turn. Select whether to display the model wire frame and whether to display the material color in the selection (the default is to display).

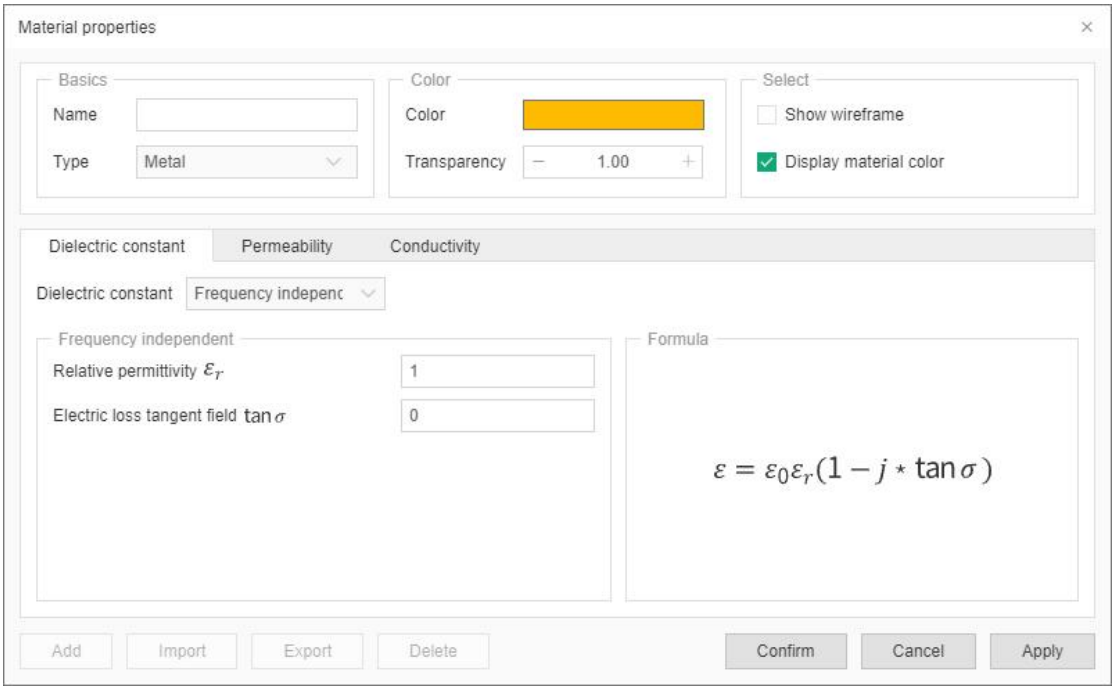


Figure 59 New material

3、 Under the dielectric constant, there are frequency dependent, input list, Debye Relaxation, Cole-Cole, Havriliak-negami and Djordjevic-Sarkar options. In each option, the user can enter different parameters to set the dielectric constant.

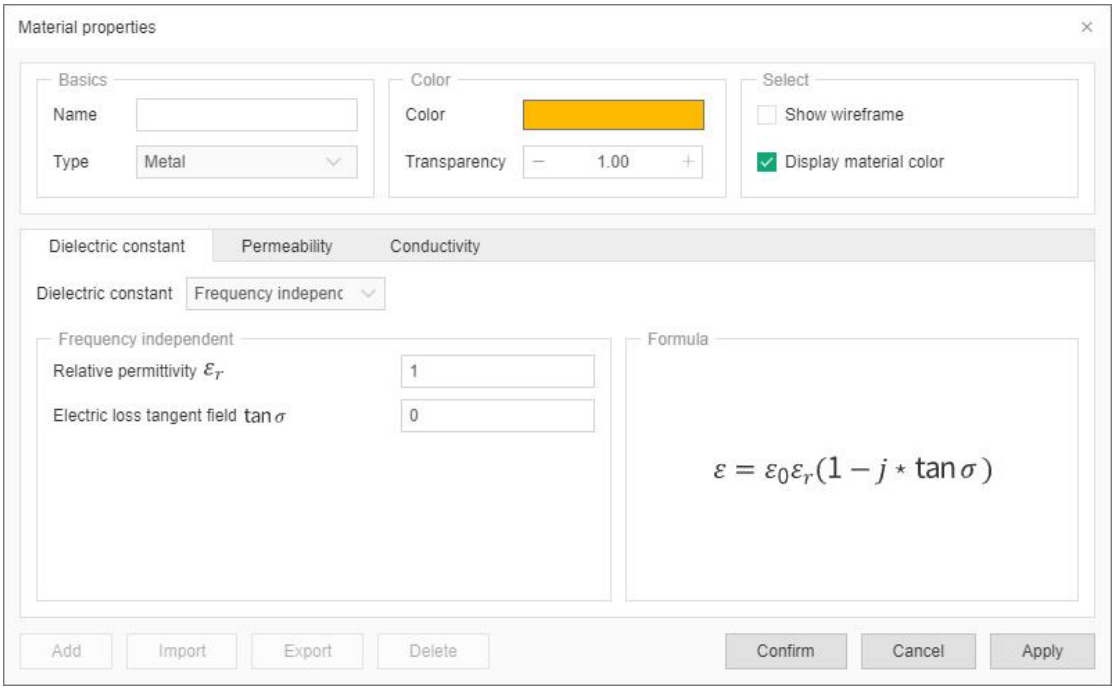


Figure 60 Dielectric constant

4、 Under magnetic permeability, there are two options: independent of frequency and input list. In each option, the user can input the parameters of magnetic permeability.

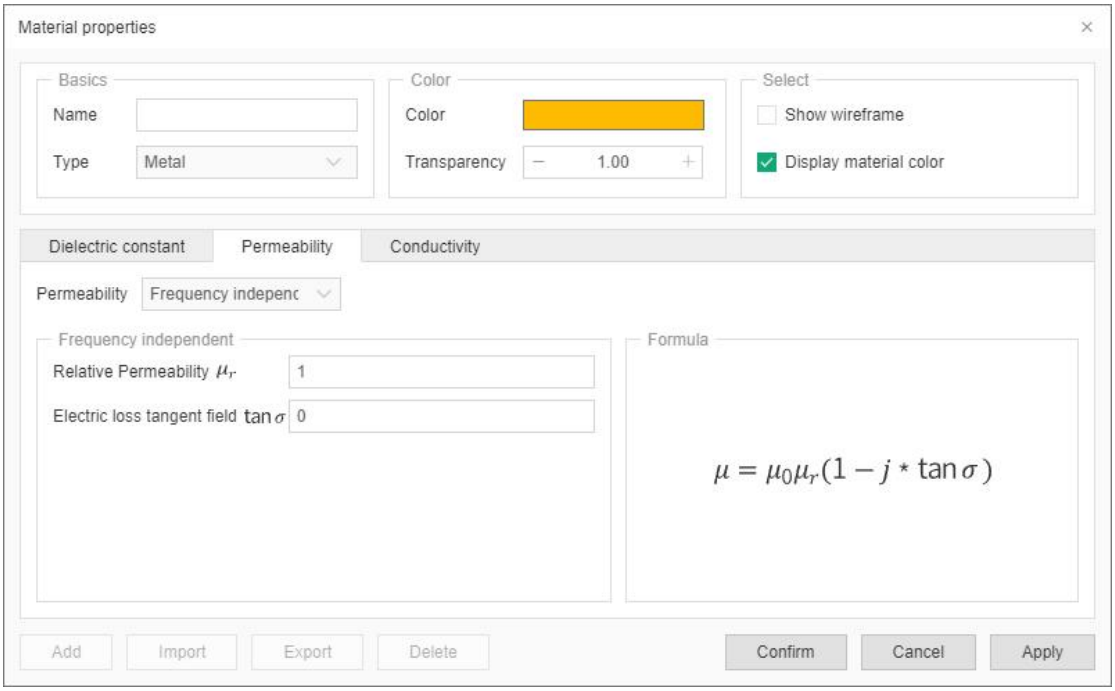


Figure 61 Permeability

5、 Under conductivity, the user can input the value of conductivity (if there is no conductivity, it can be left blank).

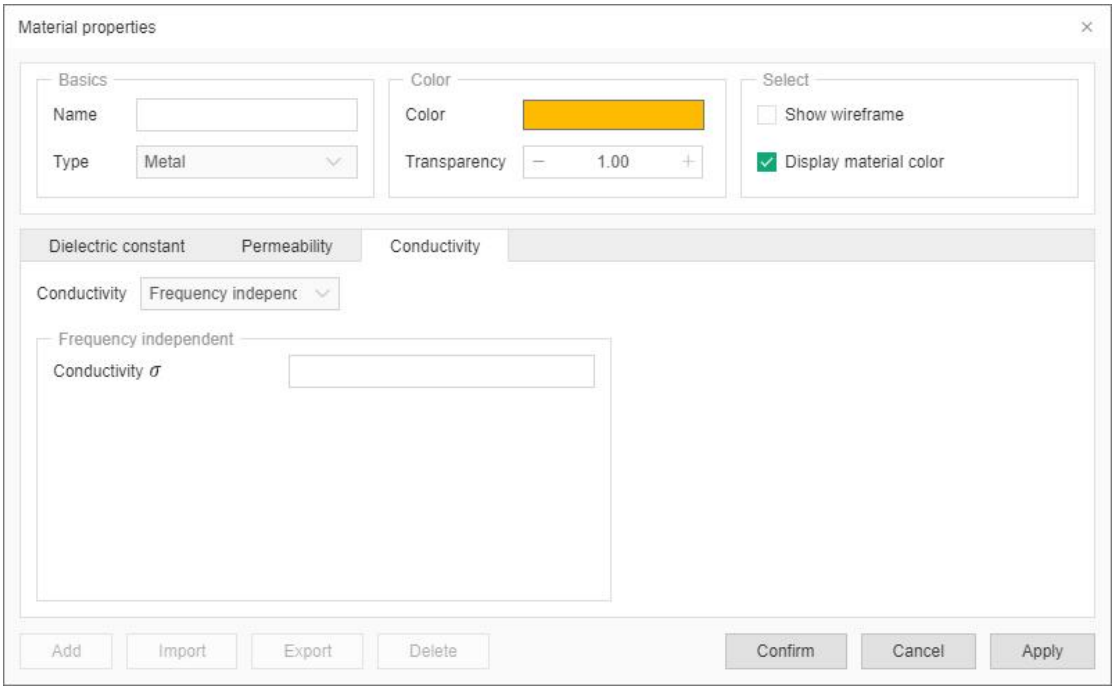


Figure 62 Conductivity

Note: If choose to enter the list, click the **Import** button to import the corresponding parameters from file.

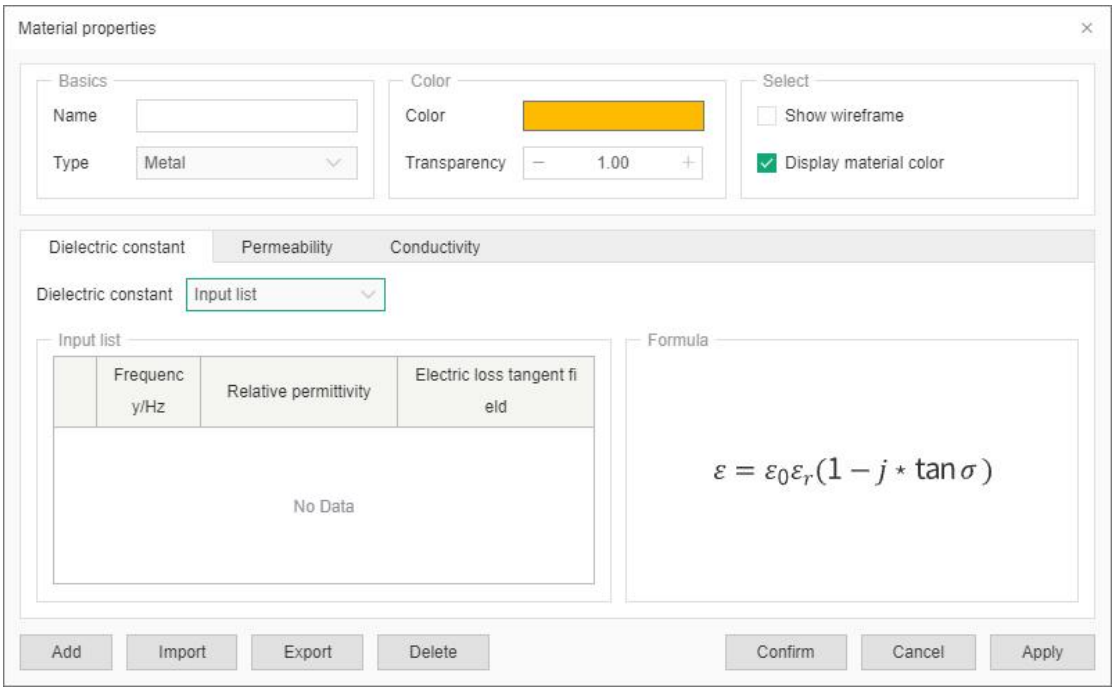


Figure 63 New material - import

3.3.3 New Anisotropic - Dielectric Material

1、 Click **New** in the material library or right-click the material button in the project manager, and click New Material.

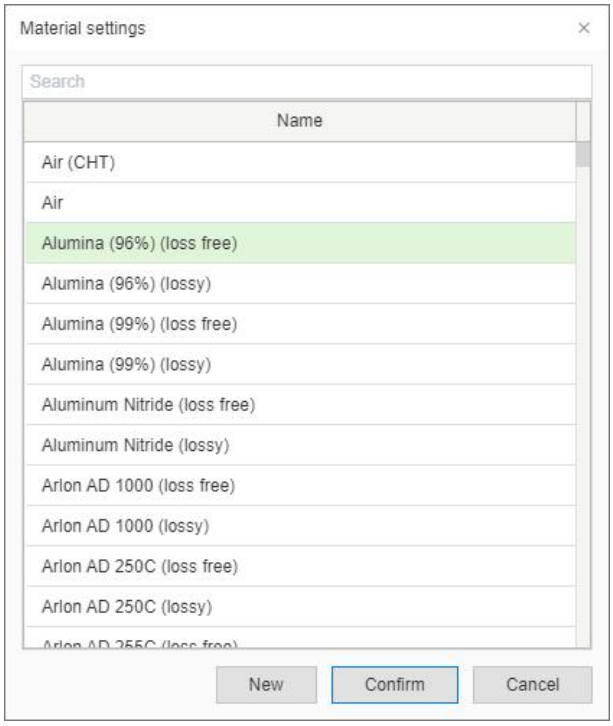


Figure 64 New material 1

2、In the new material window, select the material type as anisotropic-medium, and enter the material label name, material color and transparency in turn. Select whether to display the model wire frame and whether to display the material color in the selection (the default is display).

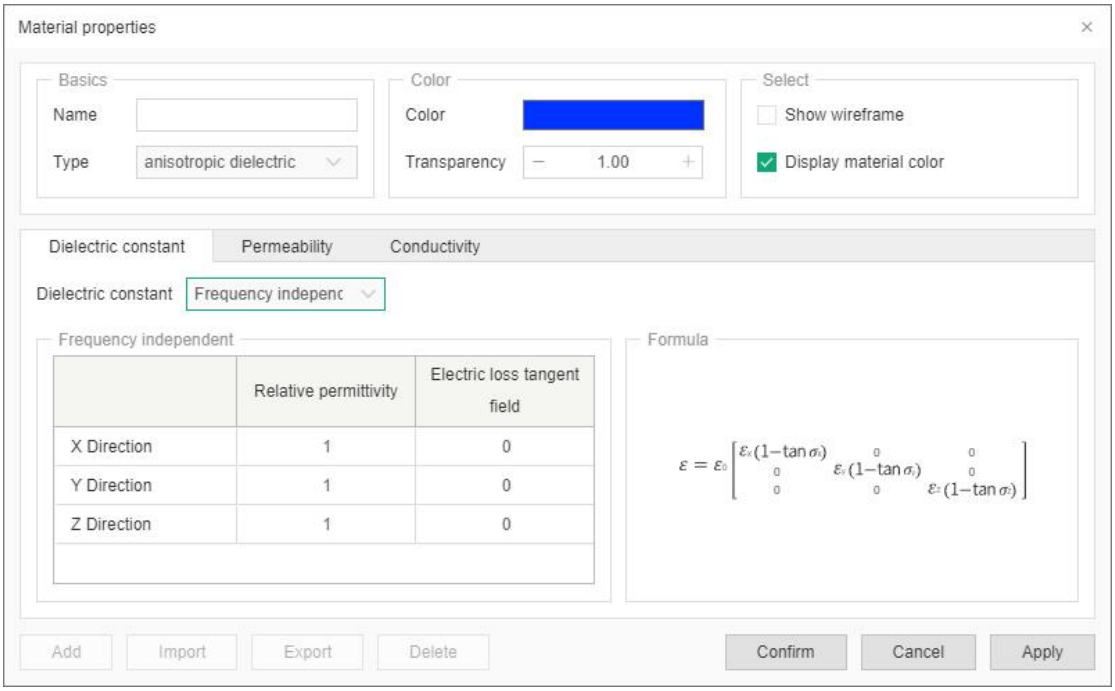


Figure 65 New material

3、 Under the dielectric constant, there are frequency-independent and input list options. In each option, the user can input different parameters to set the dielectric constant.

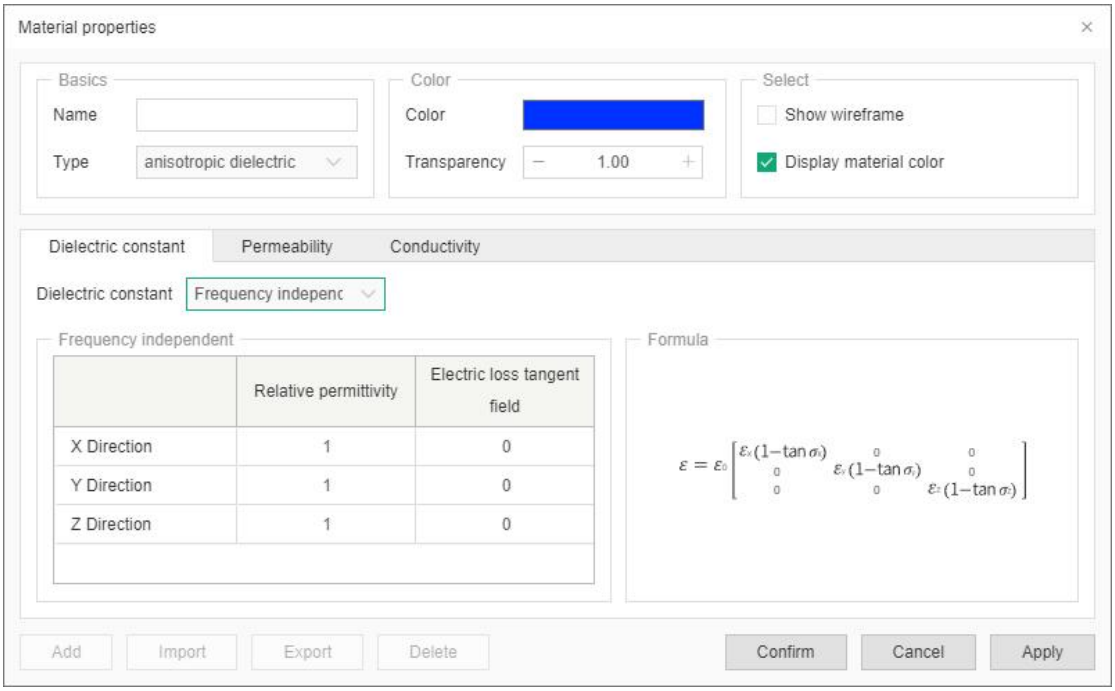


Figure 66 Dielectric constant

4、 Under magnetic permeability, there are two options: independent of frequency and input list. In each option, the user can input the parameters of magnetic permeability.

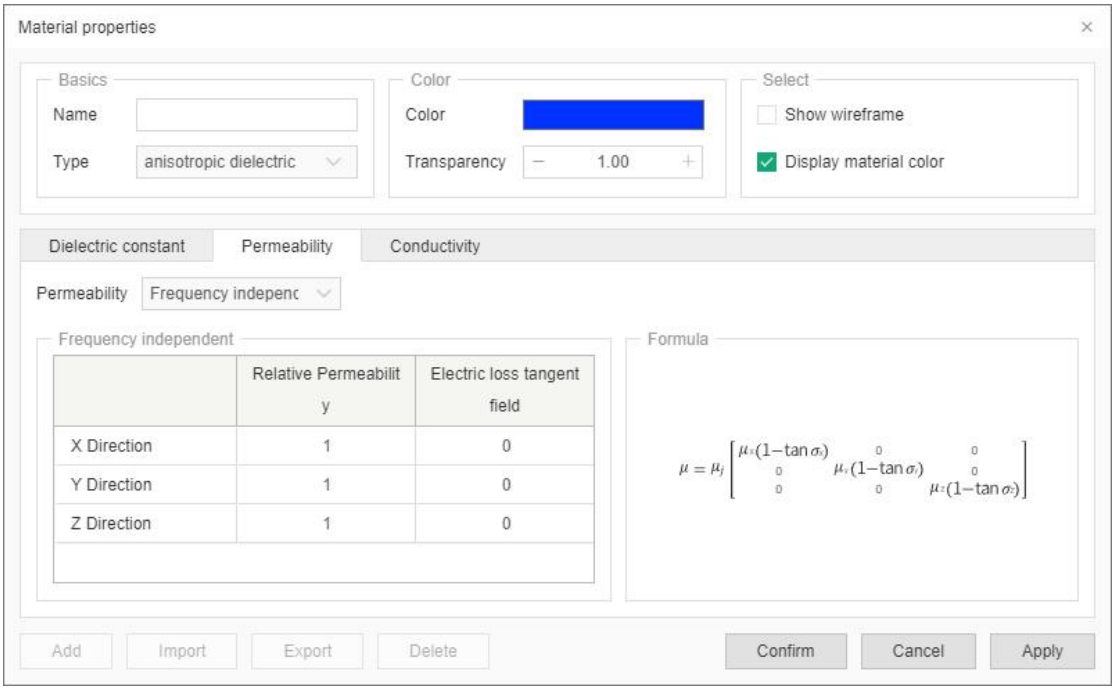


Figure 67 Permeability

5、 Under conductivity, the user can input the value of conductivity (if there is no conductivity, it can be left blank).

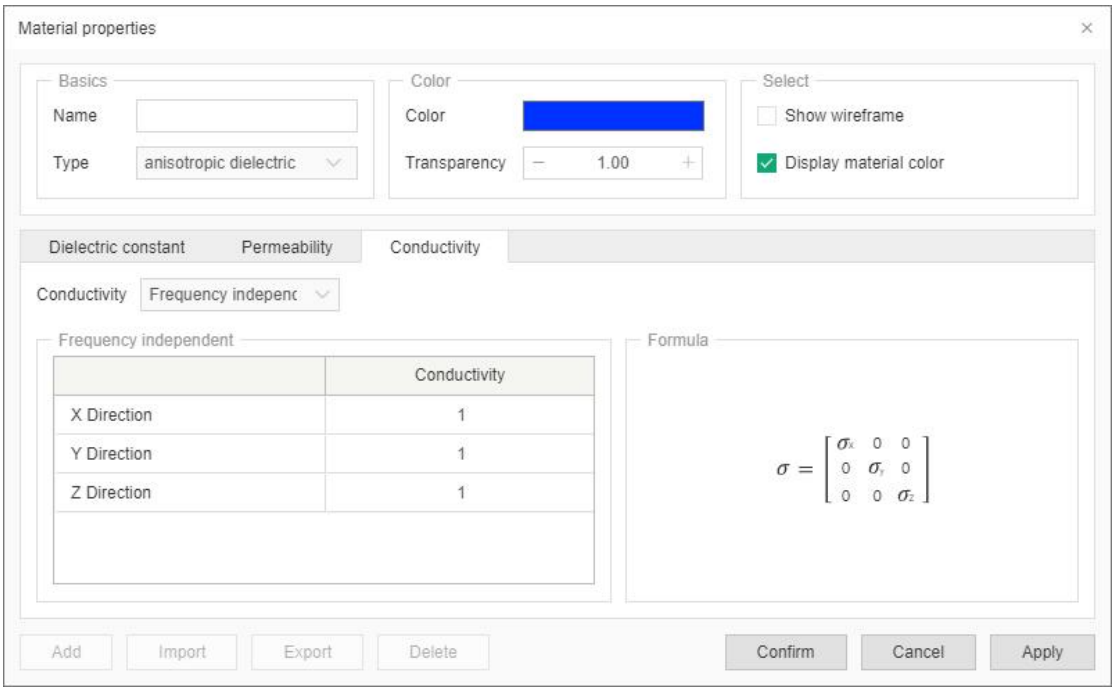


Figure 68 Conductivity

Note: If choose to enter the list, click the **Import** button to import the corresponding parameters from file.

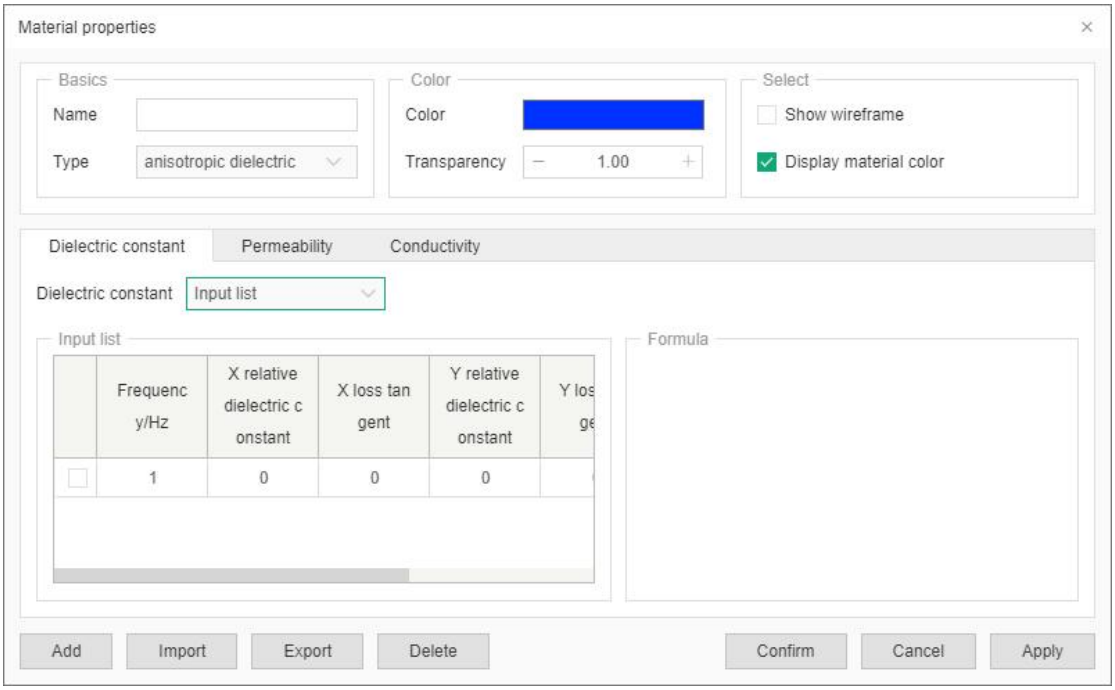


Figure 69 New material - import

3.3.4 New Anisotropic - Metal Material

1、 Click **New** in the material library or right-click the material button in the project manager, and click **New Material**.

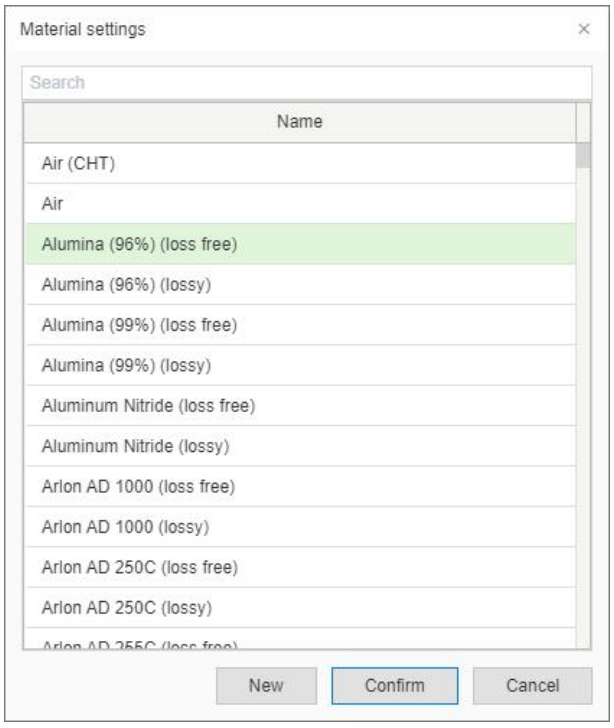


Figure 70 New material 1

2、In new material window, select the material type as anisotropic-metal, and enter the material label name, material color and transparency in turn. Select whether to display the model wire frame and whether to display the material color in the selection (the default is display).

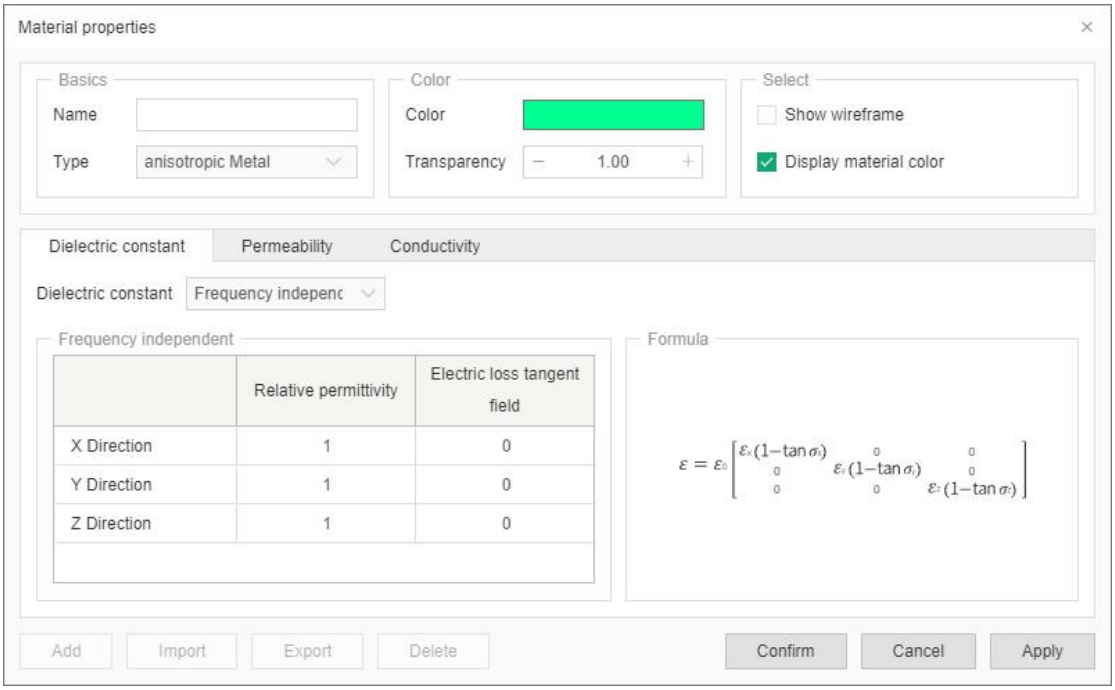


Figure 71 New material

3、 Under the dielectric constant, there are frequency-independent and input list options. In each option, the user can input different parameters to set the dielectric constant.

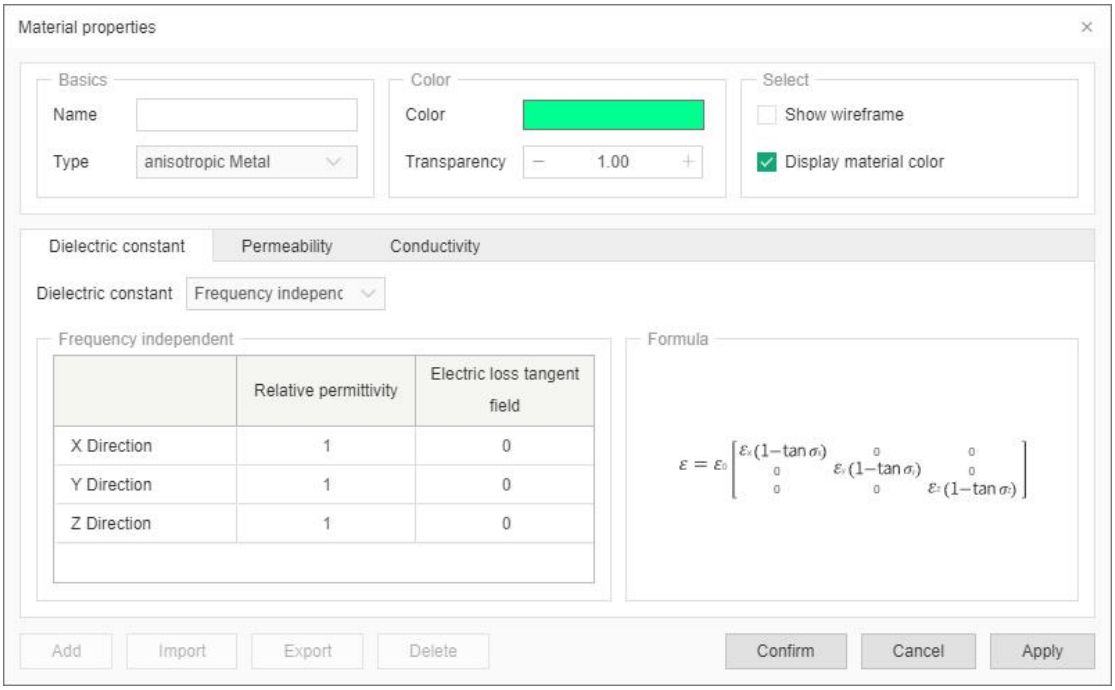


Figure 72 Dielectric constant

4、 Under magnetic permeability, there are two options: independent of frequency and input list. In each option, the user can input the parameters of magnetic permeability.

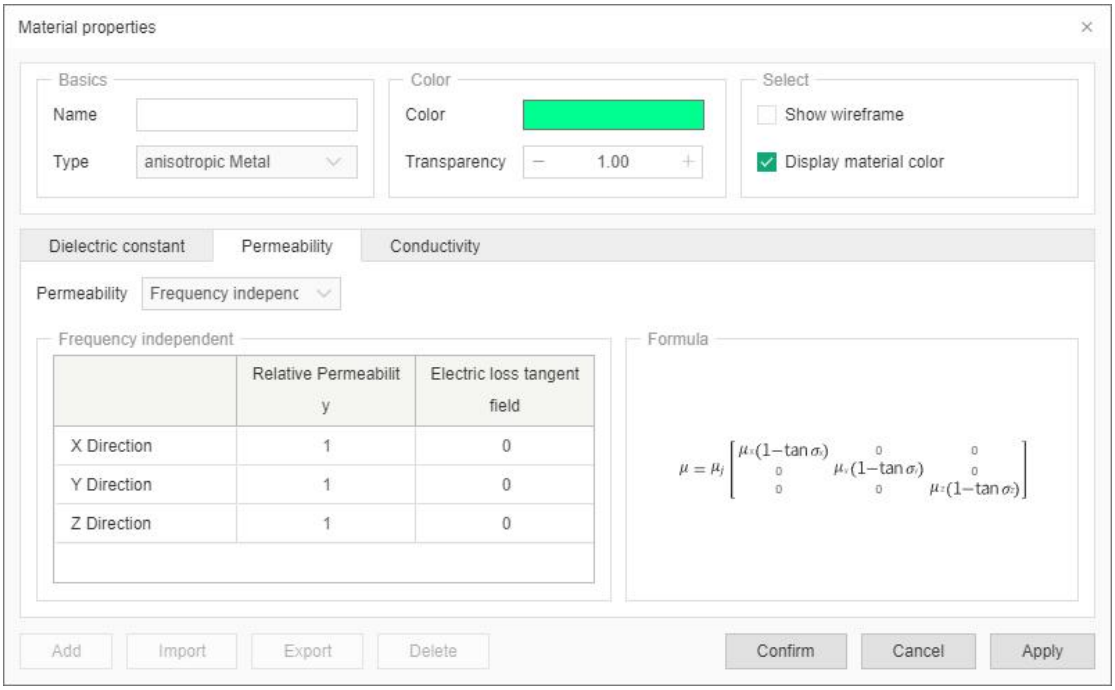


Figure 73 Permeability

5、 Under conductivity, the user can input the value of conductivity (if there is no conductivity, it can be left blank).

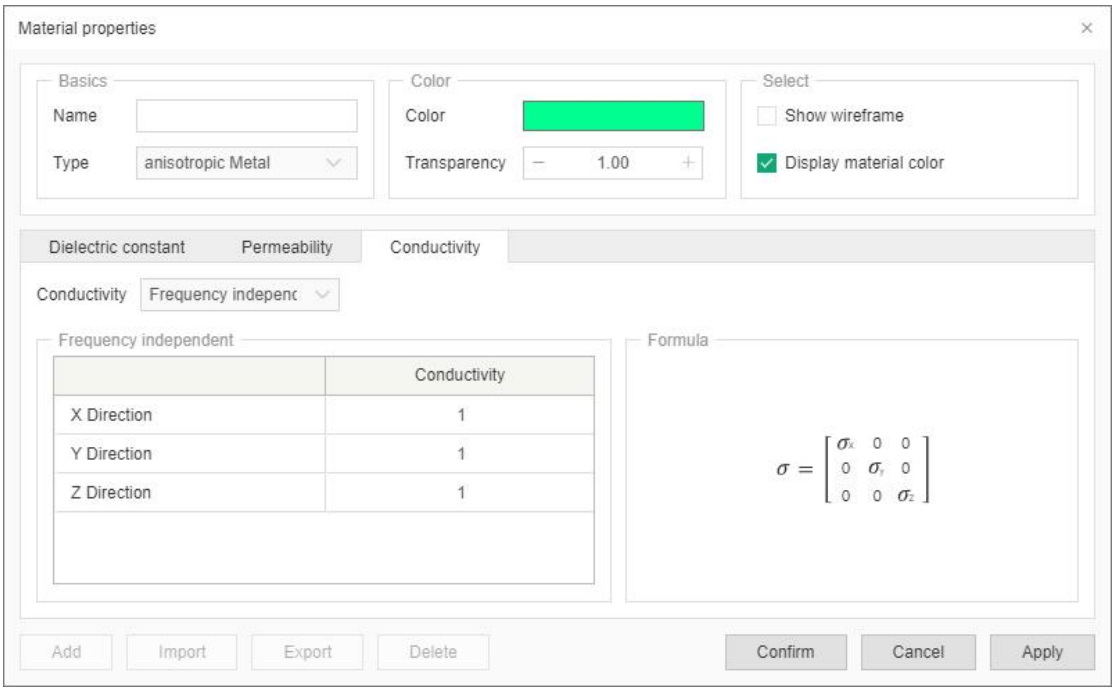


Figure 74 Conductivity

Note: If choose to enter the list, click the **Import** button to import the corresponding parameters from file.

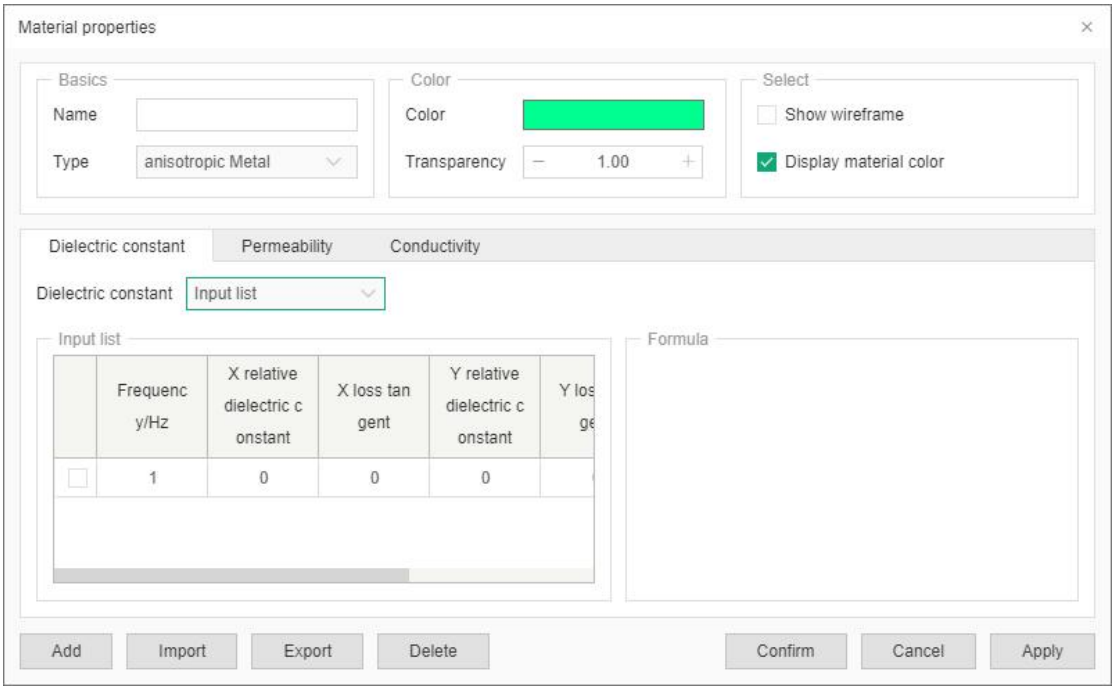


Figure 75 New material - import

3.3.5 New Impedance Condition Material

1、 Click **New** in the material library or right-click the material button in the project manager, and click **New Material.**

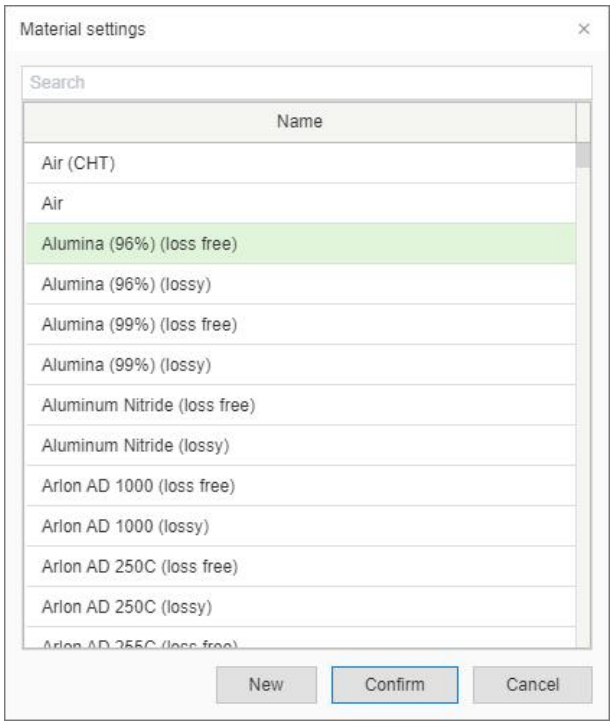


Figure 76 New material 1

2、 In the new material window, select the material type as impedance boundary condition, and enter the material label name, material color and transparency in turn. Select whether to display the model wire frame and whether to display the material color in the selection (the default is to display).

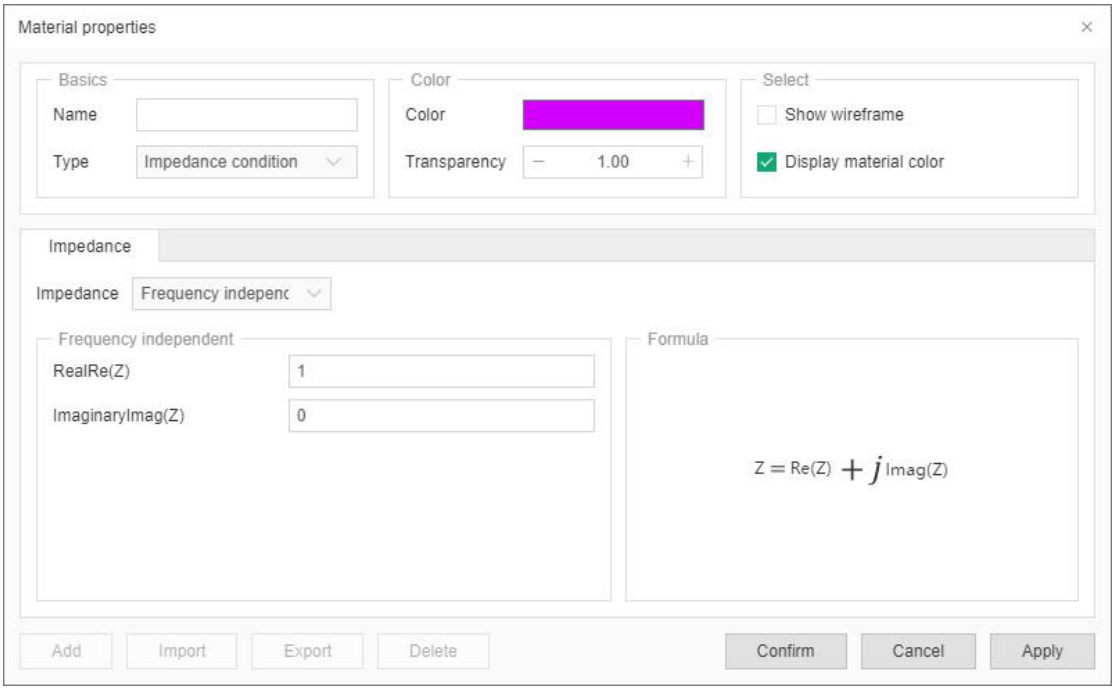


Figure 77 New material popup

3、 Under the dielectric constant, there are frequency-independent and input list options. When frequency-independent is selected, enter the real part of the impedance and the imaginary part of the impedance to set the impedance boundary condition.

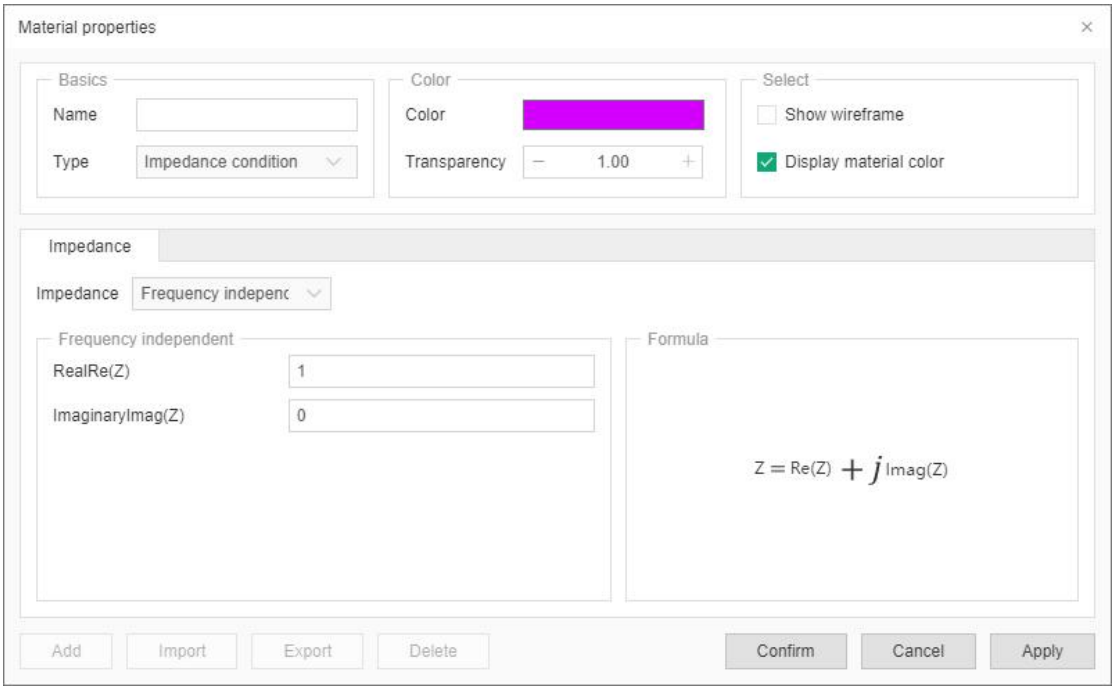


Figure 78 Dielectric constant

Note: If choose to enter the list, click the **Import** button to import the

corresponding parameters from file.

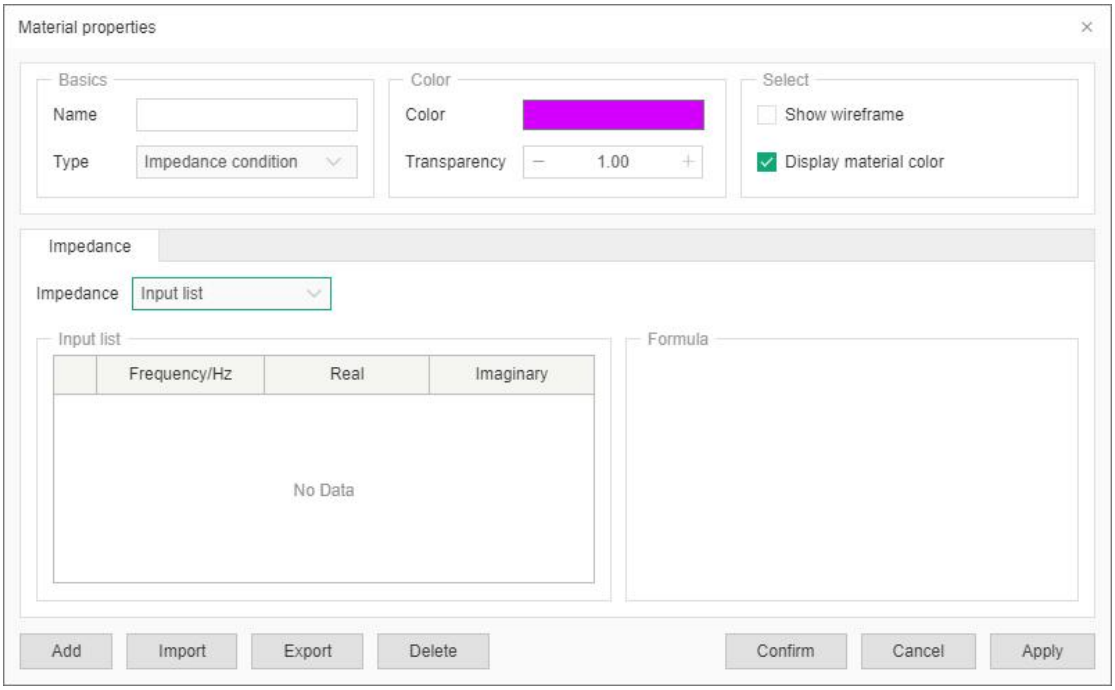


Figure 79 New material - import

3.4 Surface Model Material Settings

The surface model can only be assigned as the metal material and PEC, and the default material of the surface model is PEC. The steps of surface model material setting are:

1、 In the view area, select a certain surface model with the left mouse button, or select the label name of a certain surface model under the model manager area - calculation model - surface model tree.

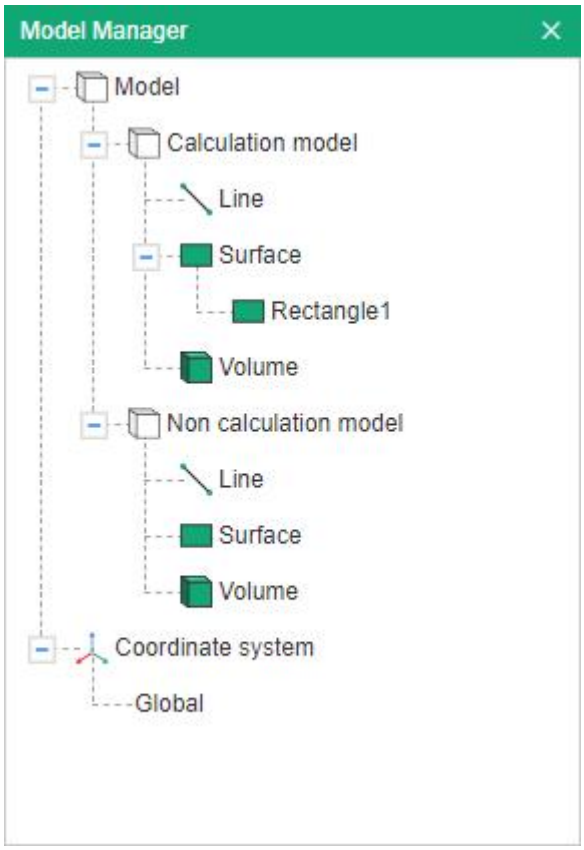


Figure 80 Surface model material settings - model manager area

2、 In the property manager area - under surface, select the label name of the surface, and click the right mouse button.

3、 Click the property label name, and the property window will pop up.

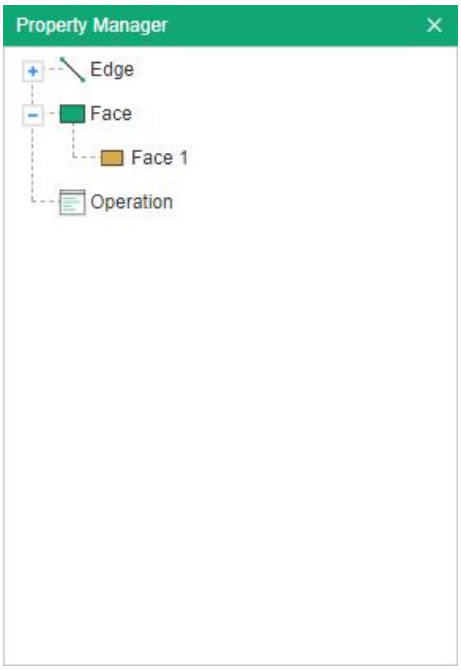


Figure 81 Surface model material setting - property manager area

4、 In the material box of the property window, click the drop-down box behind the material, click other materials, and the window of the metal material library will pop up.

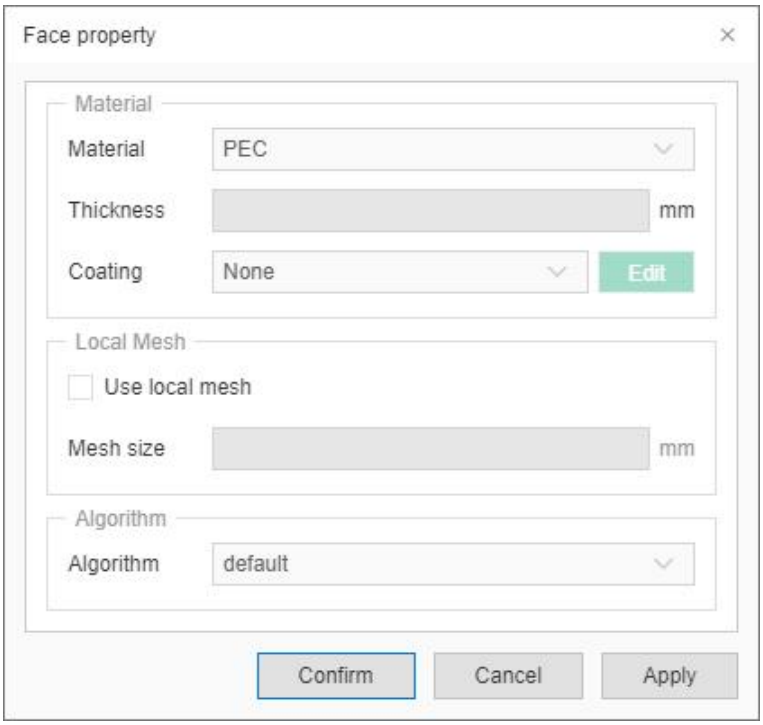


Figure 82 Surface model material settings - property pop-up window

5、 In the window of the metal material library, select the material label name, and click **Confirm**.

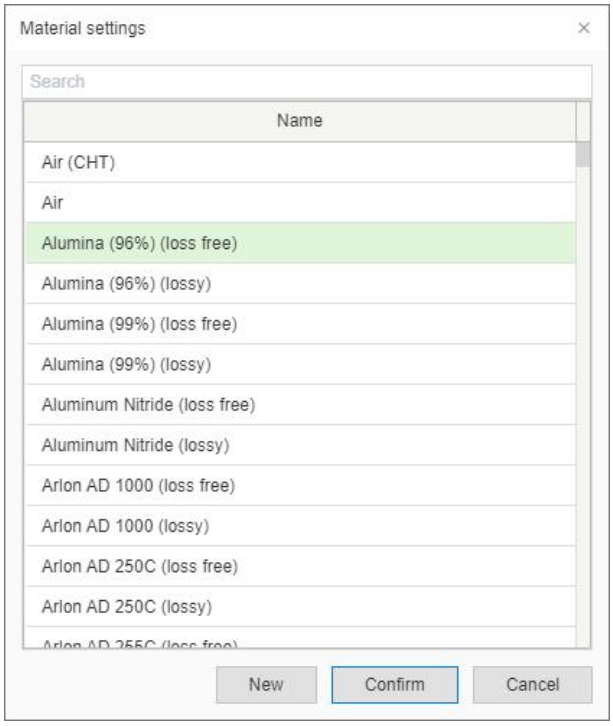


Figure 83 Surface model material settings - material window

6、 Click **Confirm** in the property window to complete the surface model material setting.

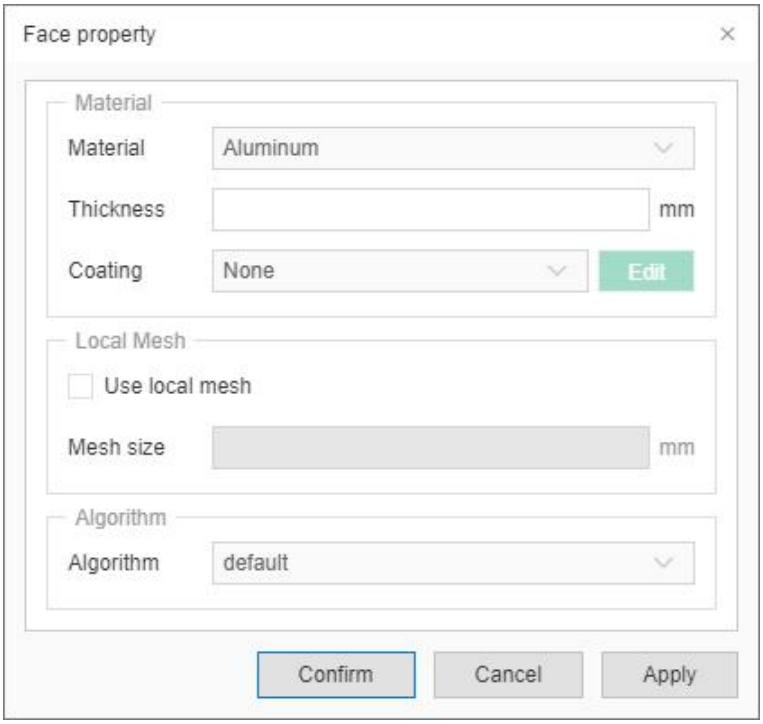


Figure 84 Surface model material settings - property window

7、 In the view area, the model color will change to the color of the material, and the label name of the material will appear under the material tree in the project manager area, and the color in front of the label name indicates the color of the material.

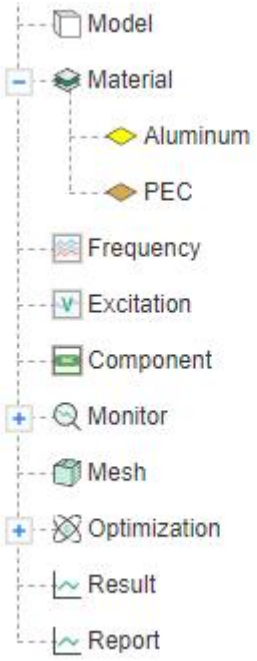


Figure 85 Surface model material setting - project manager area - material tree

3.5 Body model material settings

The body of the volume model can only be assigned as dielectric material and PEC, and the default material of the volume model is PEC. The steps of body model body material setting are as follows:

1、 In the view area, select an individual model with the left mouse button (or select the label name of an individual model under the model manager area - calculation model - solid model tree).

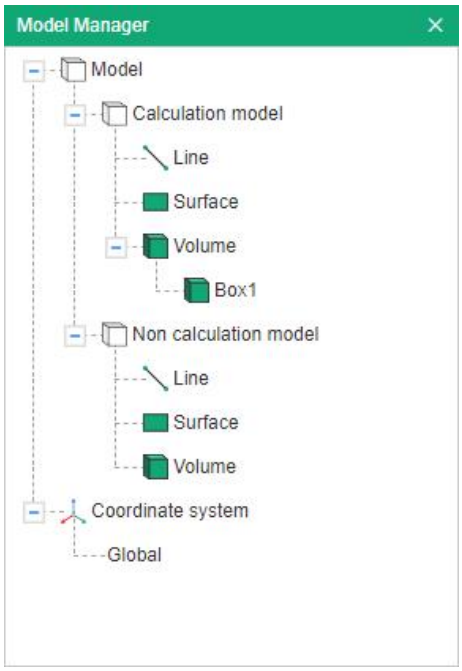


Figure 86 Volume model material settings - model manager area

2、 In the property manager area - under the body tree, select the body label name and right click.

3、 Click the property label name in the pop-up box, and the property window will pop up.

Figure 87 Solid model material setting - property manager area

4、 In the material box of the property window, click the drop-down box behind the material, click other materials, and the window of the media material library will pop up.

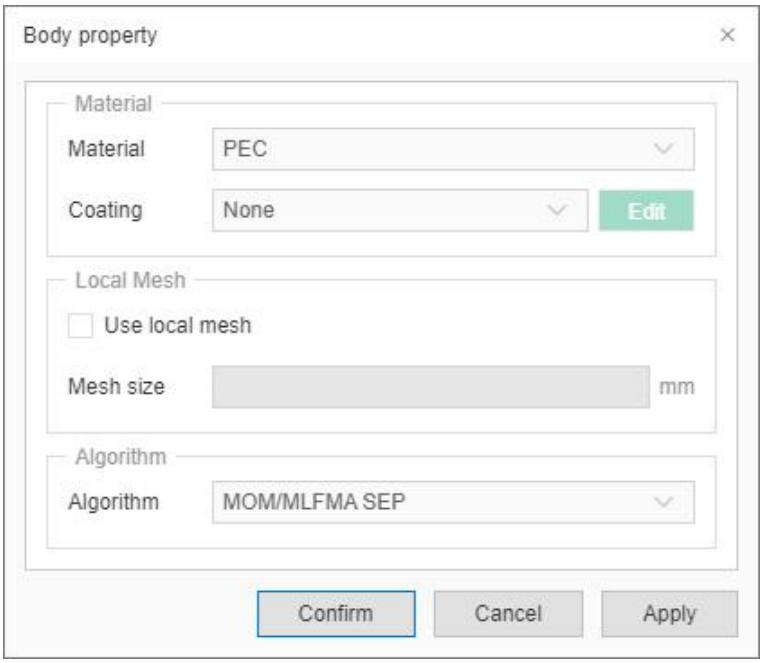


Figure 88 Volume model material settings - property pop-up window

5、 In the pop-up window of the dielectric material library, move the mouse to select the material label name, and click **Confirm** to complete the material selection.

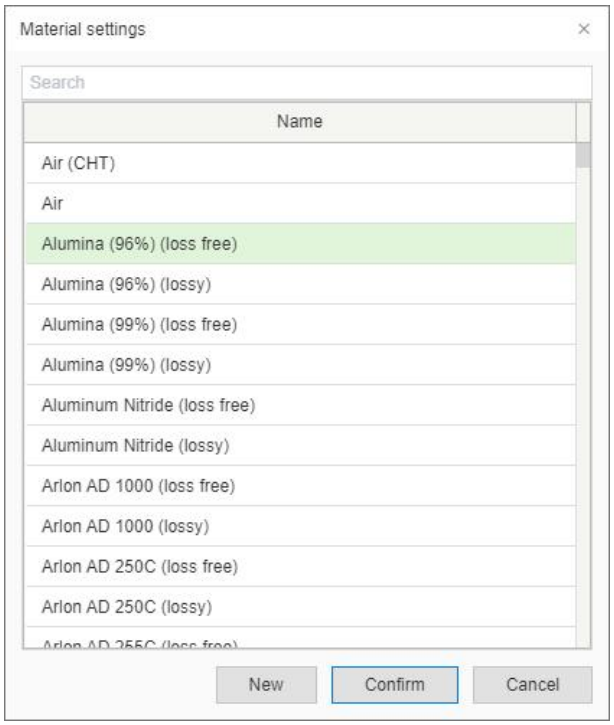


Figure 89 Body model materiasl settings - material window

6、 Click **Confirm** in the property window to complete the volume model material setting.

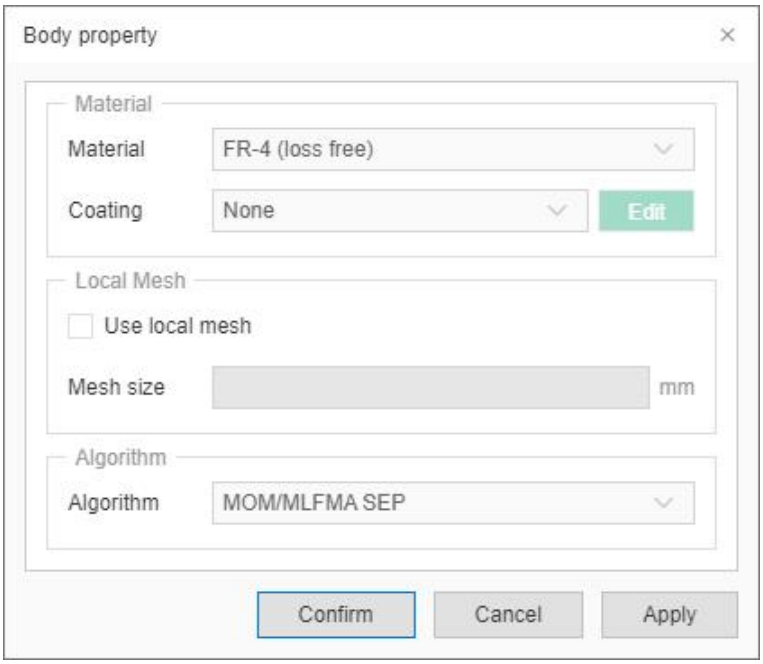


Figure 90 Volume model material settings - property

7、 In the view area, the model color changes to the color of the material, and the label name of the material will appear under the material tree in the project manager area, and the color in front of the label name indicates the color of the material.

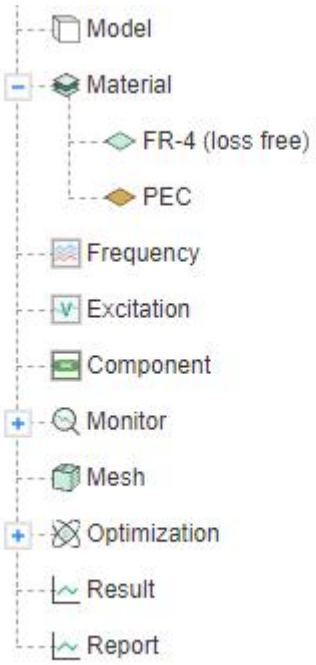


Figure 91 Solid model material setting - project manager area - material tree

3.6 Material settings for faces in body models

Generally, the material of surfaces on a volume model is the material of the volume model itself. However, the material of surfaces on a volume model can also be defined as metal materials (if so, the thickness of the surface also needs to be defined) or PEC. The steps to set the material of the surface in the solid model are as follows:

1、 In the view area, select an individual model with the left mouse button (or select the label name of an individual model under the model manager area - calculation model - solid model tree).

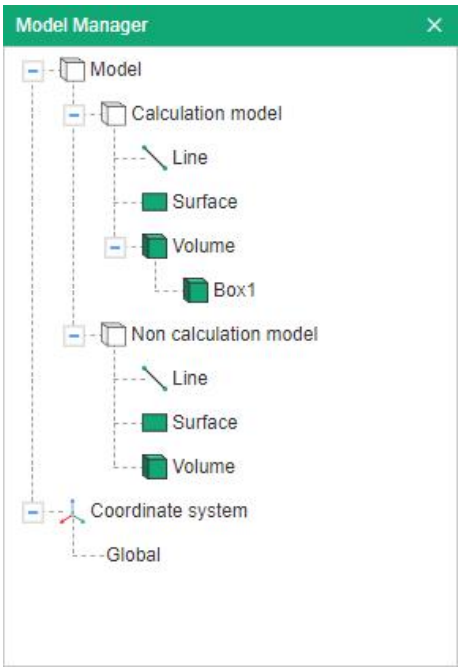


Figure 92 Material Settings for Faces in Volume Models - Model manager Area

2、 In the property manager area - under the surface tree, select the signature, and click the right mouse button (note: the label name here can be selected multiple times).

3、 Click the property label name in the box, and the property window will pop up.

Figure 93 Surface Material Settings in Volume Model - Property manager Area

4、 In the material box of the property window, click the drop-down box behind the material, click other materials, and the window of the metal material library will pop up.

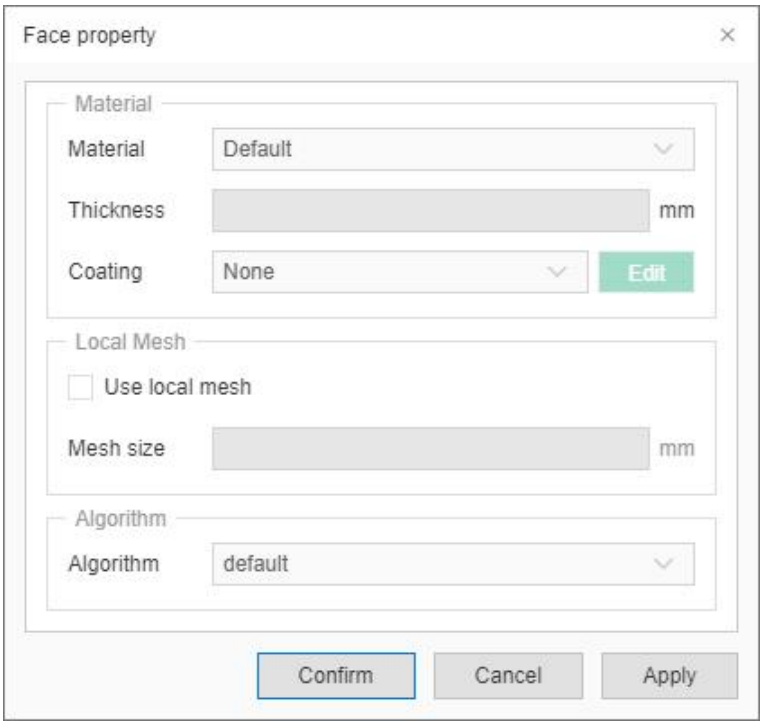


Figure 94 Material settings for faces in volume models - property popup

5、 In the window of the metal material library, move the mouse to select the material label name, and click **Confirm** to complete the material selection.

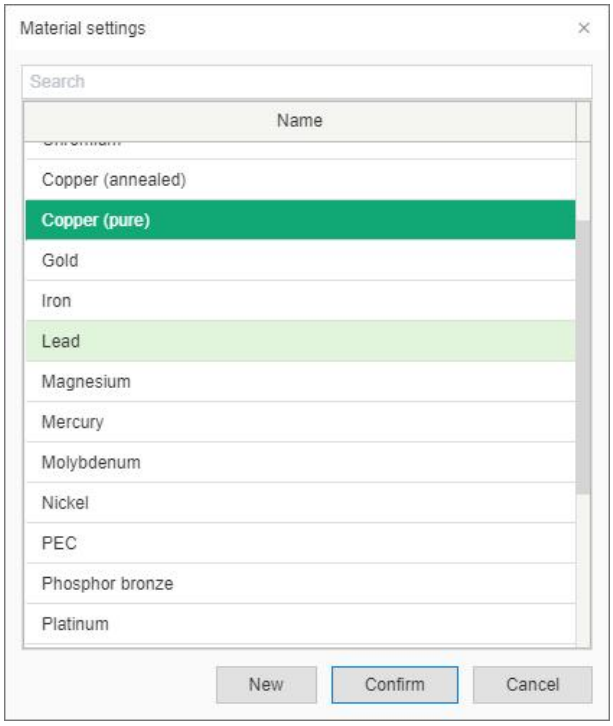


Figure 95 Body model material settings - material window

6、 Click **Confirm** in the property window to complete the material setting of the surface in the volume model.

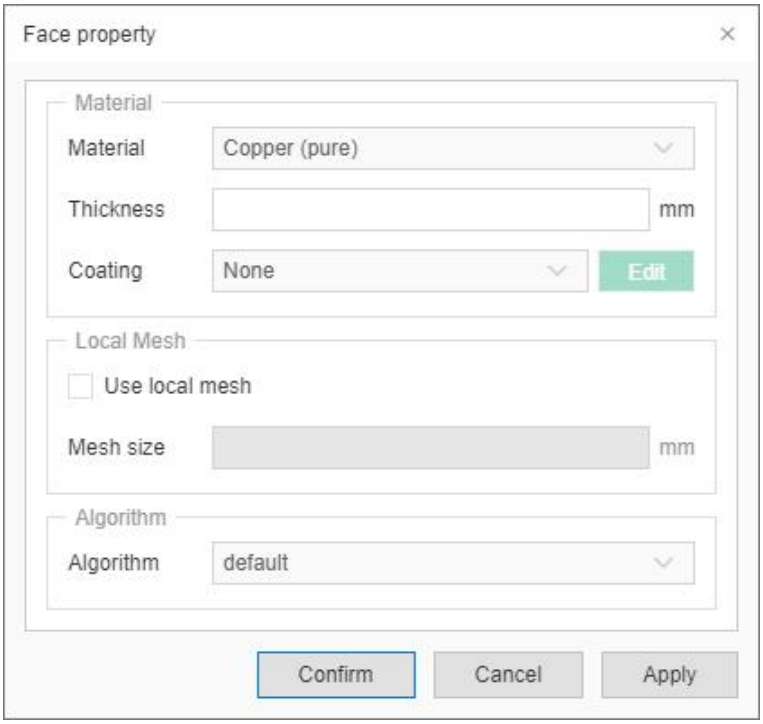


Figure 96 Material settings for faces in volume models - property

7、 In the view area, the color of the surface of the solid model changes to the color of the material, and the label name of the material will appear under the material tree in the project manager area, and the color before the label name indicates the color of the material.

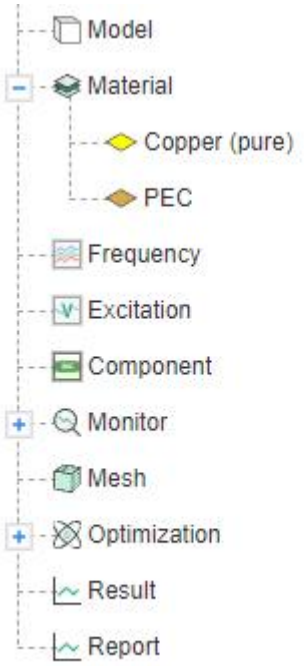


Figure 97 Material setting of surface in solid model - project manager area - material tree

3.7 Coating material

The coating material option adds a material structure with thickness on the surface of an existing model. After the coating material is selected, the material properties and thickness of the coating need to be defined. The setting of the coating material includes coating material on solid models and coating material on surface models.

3.7.1 Coating material settings on model surfaces

In the view area, select a certain surface model (left mouse button), or select the label name of a certain surface model under the model manager area - calculation model - surface model tree.

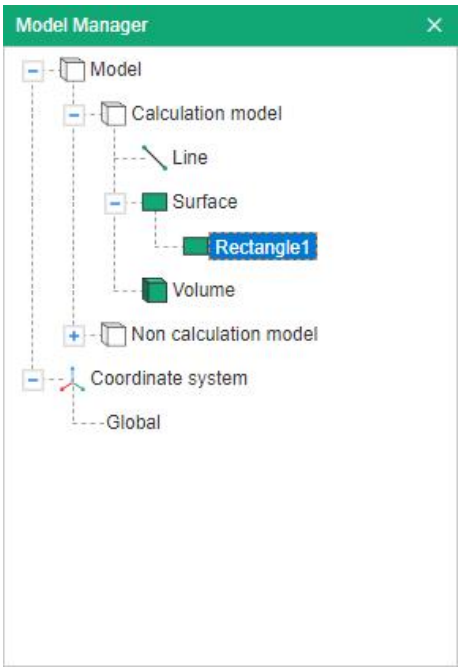


Figure 98 Surface Model Material Settings - Model manager Area

1、 In the property manager area - under surface, select the label name of the surface, and click the right mouse button.

2、 Click the property label name in the pop-up box, and the property window will pop up.

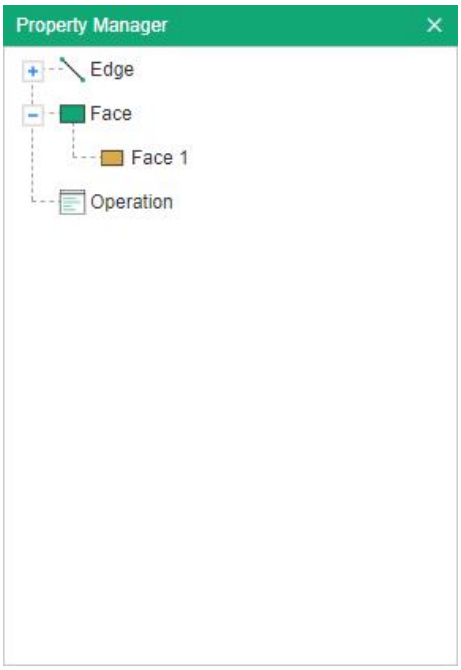


Figure 99 Surface model material setting - property manager area

3、 In the material box of the property window, click the drop-down box behind the coating and click Enable.

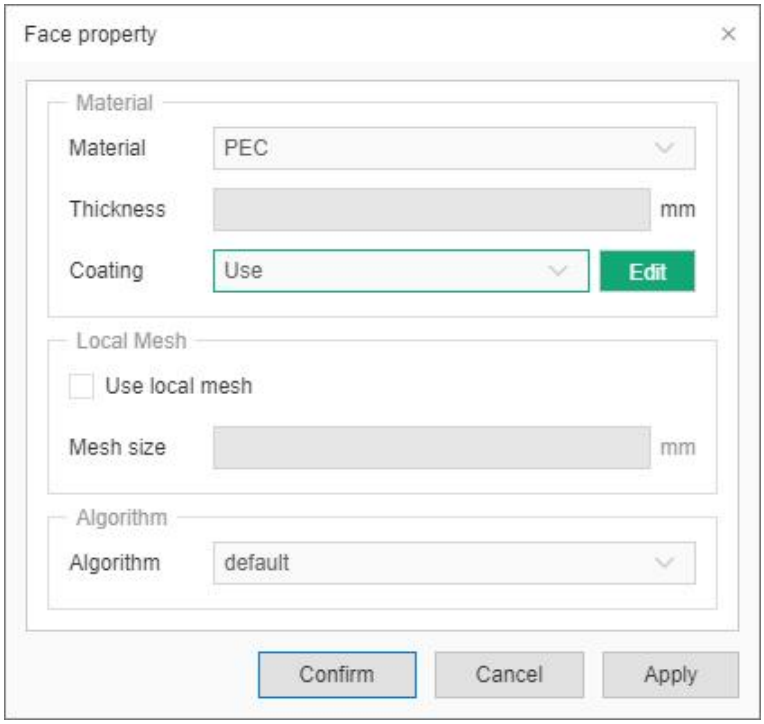


Figure 100 Surface model material settings - property window

4、 Edit the coating, select the material name, input the thickness, and click **Confirm** to complete the coating setting.

5、 Click **Confirm** in the property window to complete the surface model material setting.

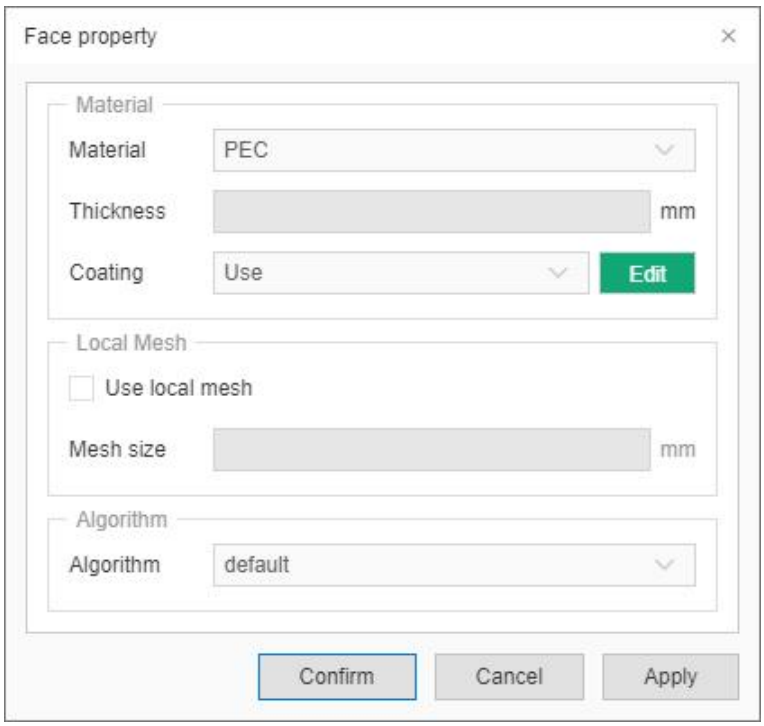


Figure 101 Surface model material settings - property window

3.7.2 Coating material settings for volume model surfaces

In the view area, select an individual model with the left mouse button, or select the label name of an individual model under the model manager area - calculation model - solid model tree.

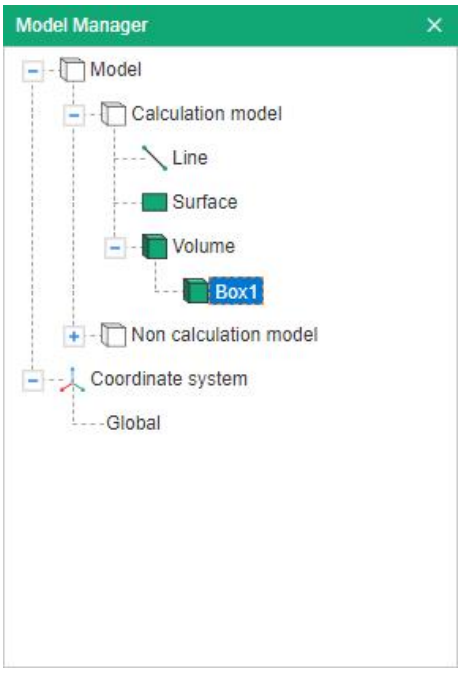


Figure 102 Volume model material settings - model manager area

1、 In the property manager area - volume model, select the label name of the volume, and click the right mouse button.

2、 Click the property label name in the pop-up box, and the property window will pop up.

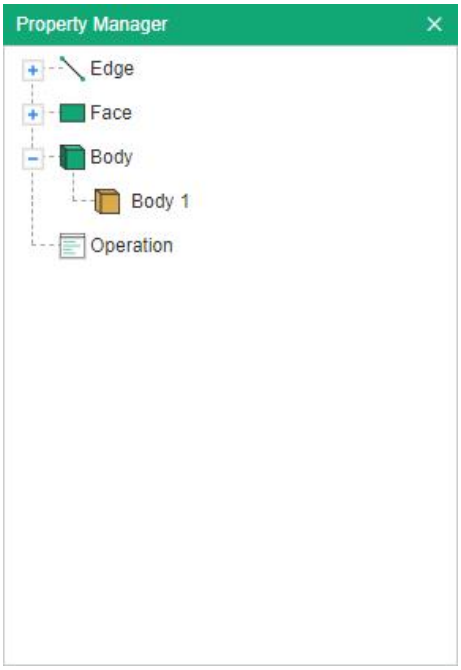


Figure 103 Solid model material setting - property manager area

3、 In the material box of the property window, click the drop-down box behind the coating and click use.

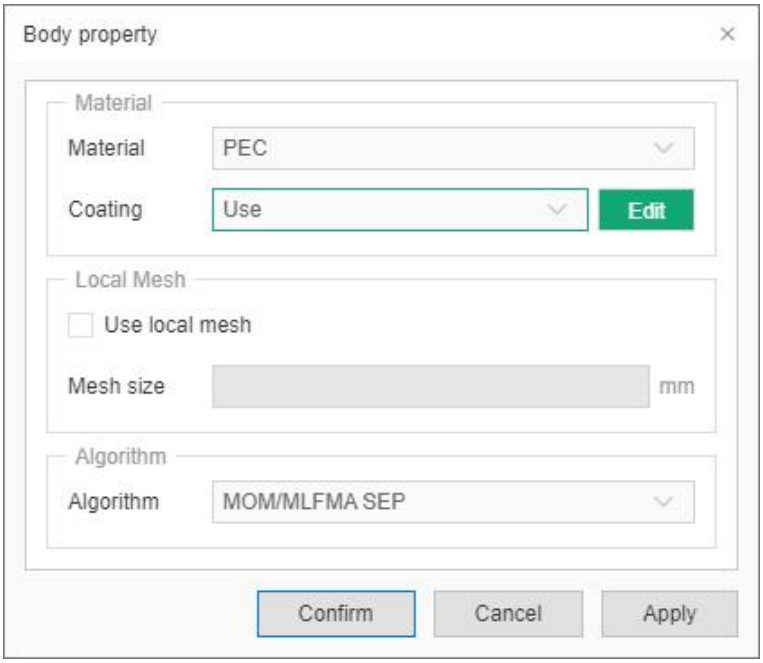


Figure 104 Volume model material settings - property window

4、 Edit the coating, select the material name, input the thickness, and click **Confirm** to complete the coating setting.

5、 Click **Confirm** in the property window to complete the surface coating material setting of the volume model.

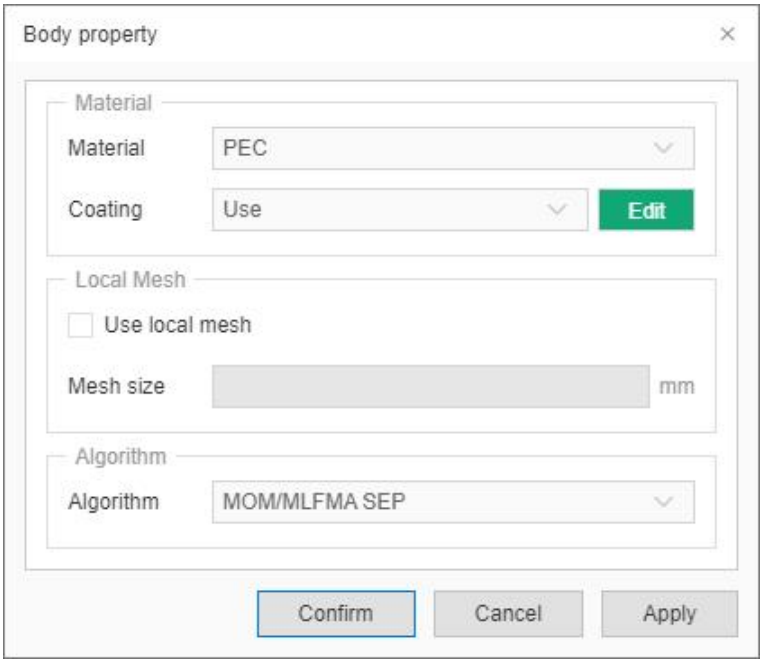


Figure 105 Volume model material settings - property window

4 View Module

The view module controls the display and interaction of models and the settings of the software interface, which includes three-view, view translation, rotation, zoom, self-adaptation, model display/hide, length and angle measurement, cross-section settings etc.

4.1 Orient

To realize the three-view operation and the translation and rotation operation of the view.

4.1.1 Orient

Orient operations include **Isometric View, Top View, Bottom View, Front View, Back View, Left View** and **Right View**.

Button	Button Label Name	Function
	Isometric View	Display the perspective view of the model
	Top View	Display the top view of the model
	Bottom View	Display the bottom view of the model
	Front View	Display the front view of the model
	Back View	Display the back view of the model
	Left View	Display the left view of the model
	Right View	Display the right view of the model

4.1.2 Move/Rotate

Move and **Rotate** operations can realize the translation and rotation of the viewing area.

Button	Button Label Name	Function
	Move	Implement moving of the view

	Rotate	Implement the rotation of the view

4.1.3 Zoom

Zoom operations include **Zoom In**, **Zoom Out**, **Fit All** and **Fit**.

Button	Button Label Name	Function
	Zoom In	Enlarge the view
	Zoom Out	Realize the zoom out of the view
	Fit	Selected models in the modeling view area are displayed as a reasonable size
	Fit All	Realize that all models in the modeling view area are displayed at a reasonable size

4.2 Tool

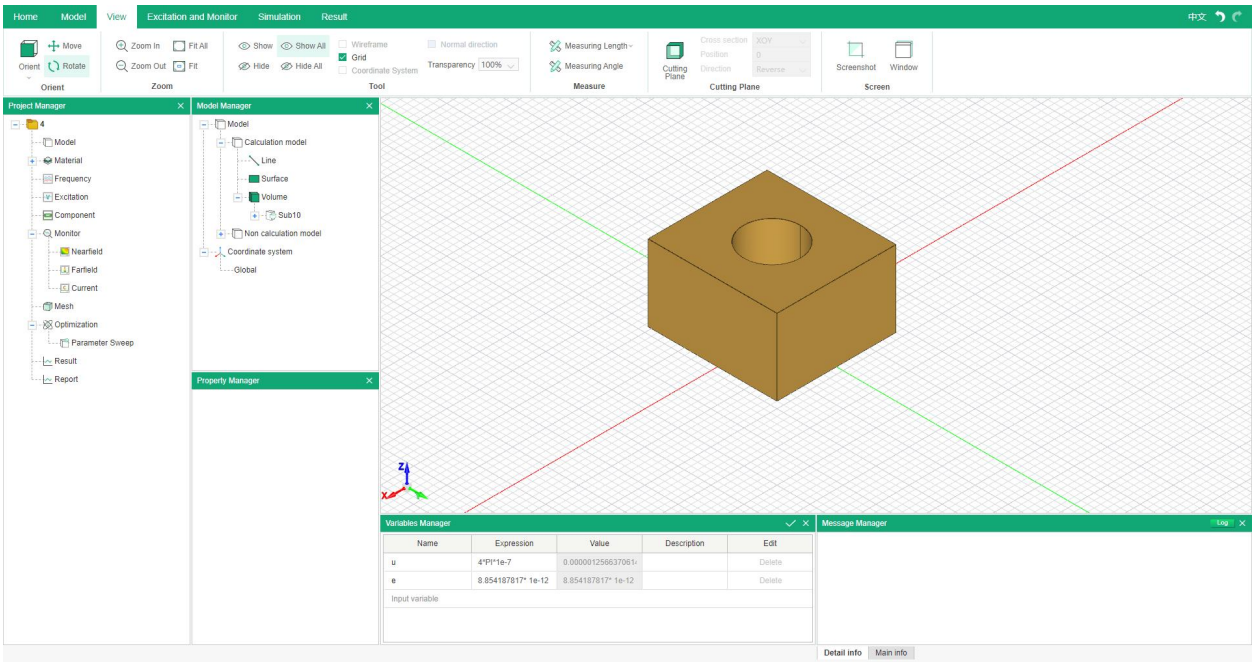
4.2.1 Show

Display part includes **Display, Hide, Show All,** and **Hide All**.

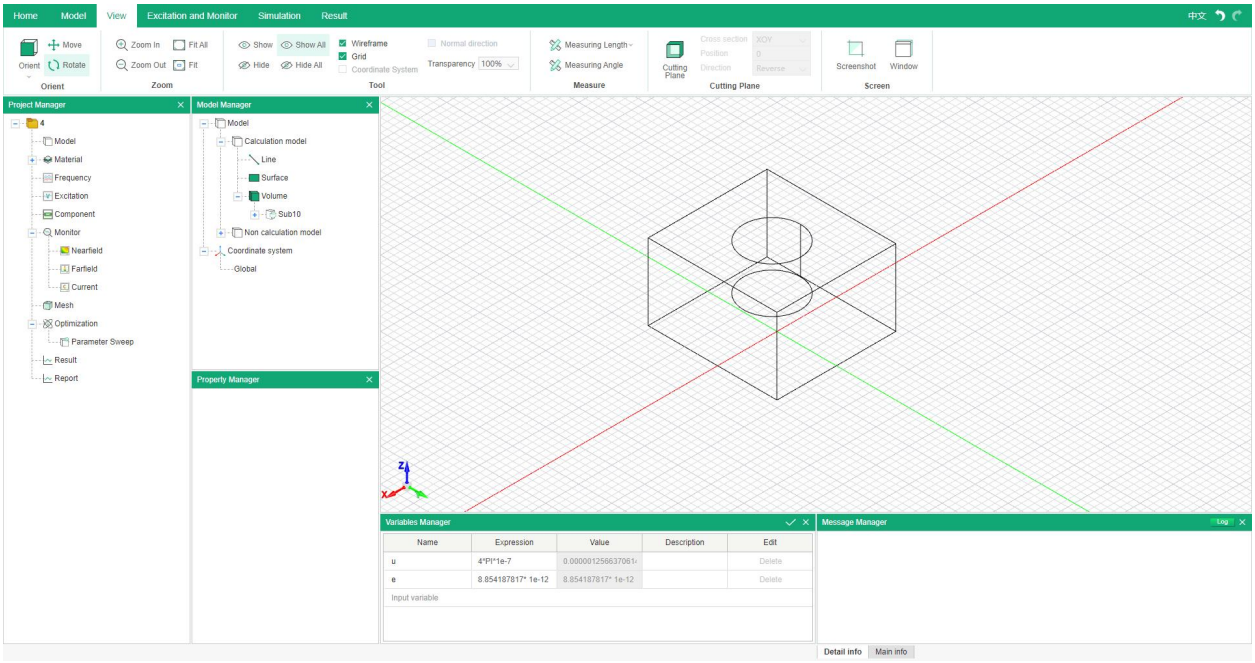
Button	Button Label Name	Function
	Show	Display the selected model in the modeling view area
	Hide	Hide the selected model in the modeling view area
	Show All	Display all models in the modeling view area
	Hide All	Hide all models in the modeling view area

4.2.2 Wireframe

As the **Wireframe** is checked, the view interface displays model wire frame.



(a) Before ticking the model wire frame

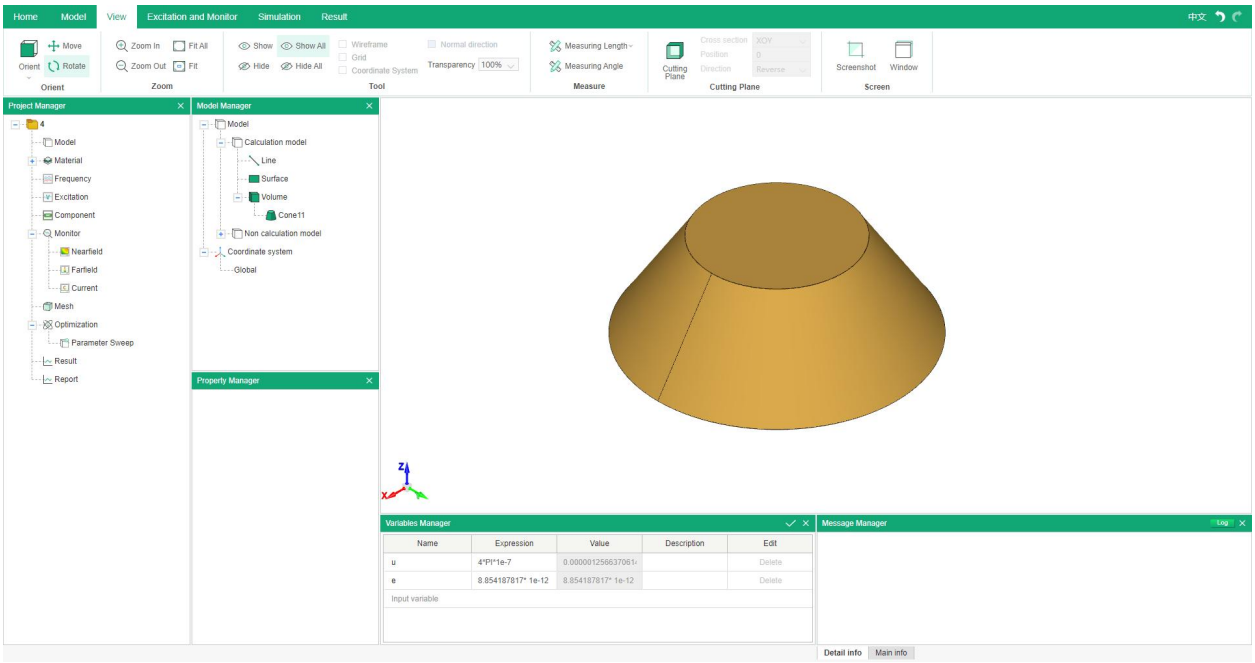


(b) After checking the model wire frame

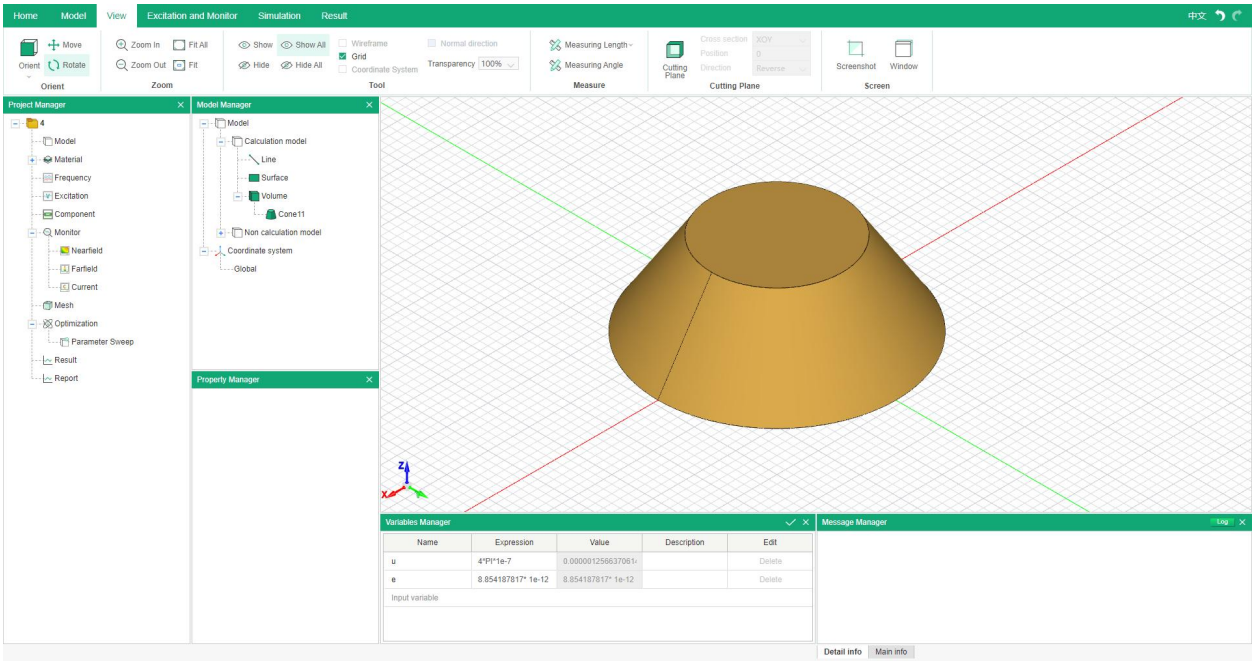
Figure 106 Tick model wire frame

4.2.3 Grid

As **Grid** is checked, the view interface displays the modeling reference grid.



(a) Check before grid display

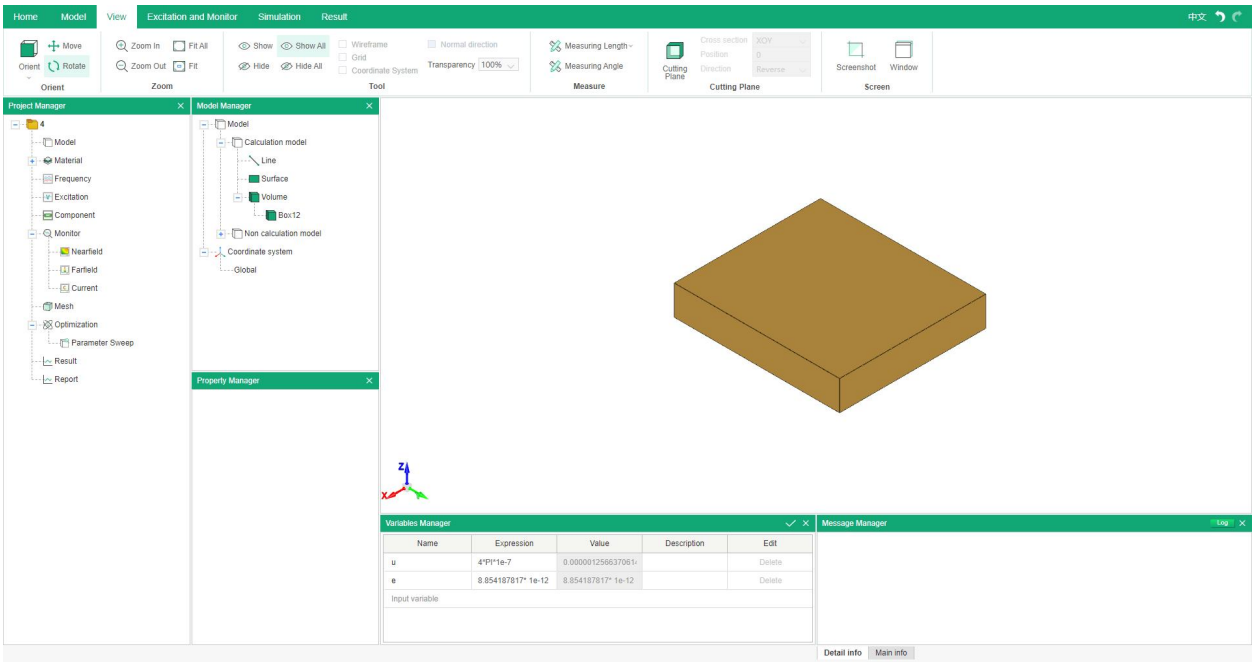


(b) After checking the grid display

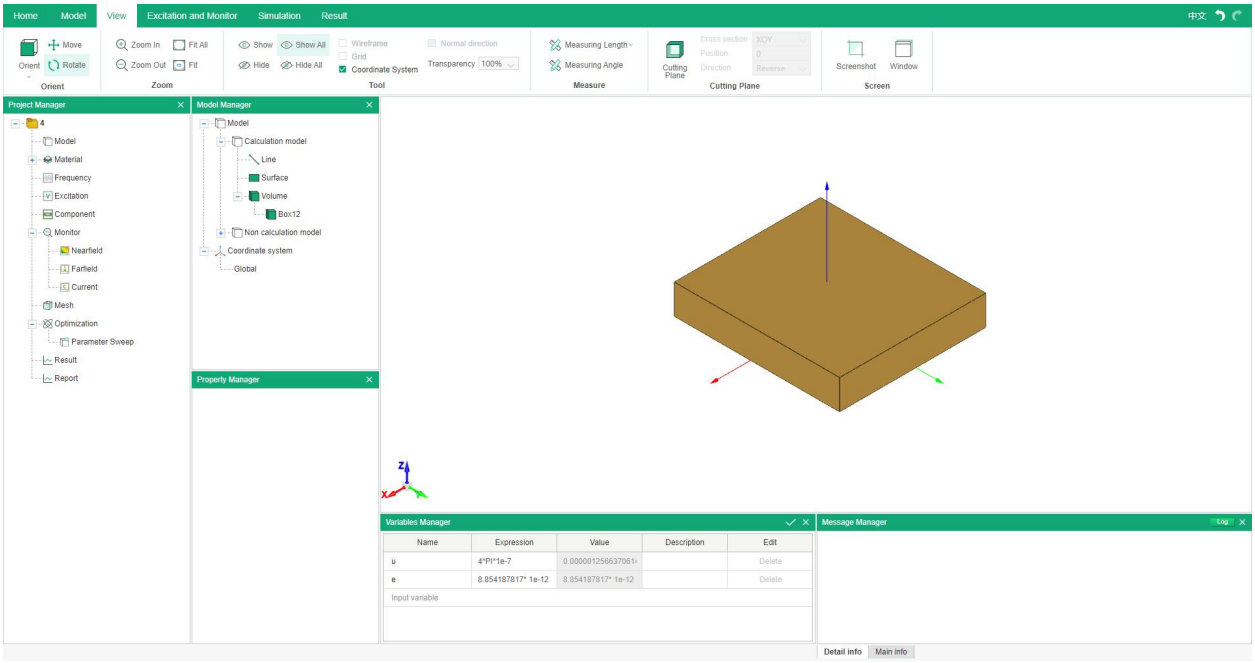
Figure 107 Check grid

4.2.4 Coordinate System

As **Coordinate System** is checked, the view interface displays the coordinate system.



(a) Before checking the coordinate system display



(b) After checking the coordinate system display

Figure 108 Check the coordinate system

4.2.5 Transparency

In **Transparency**, the transparency of the model can be set up from 0% to 100%.

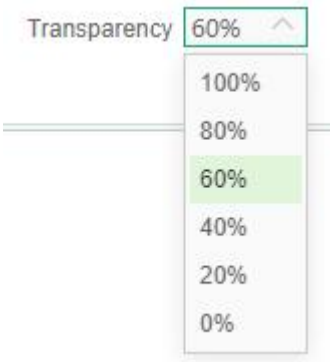


Figure 109 Transparency settings

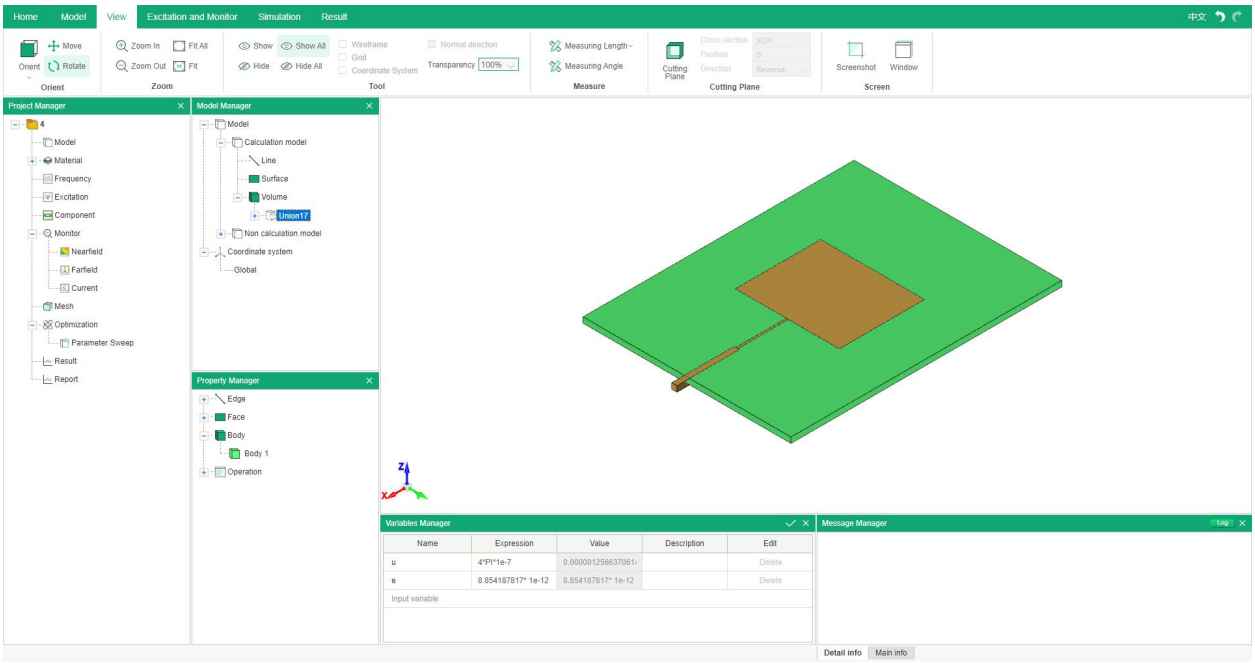


Figure 110 Original model

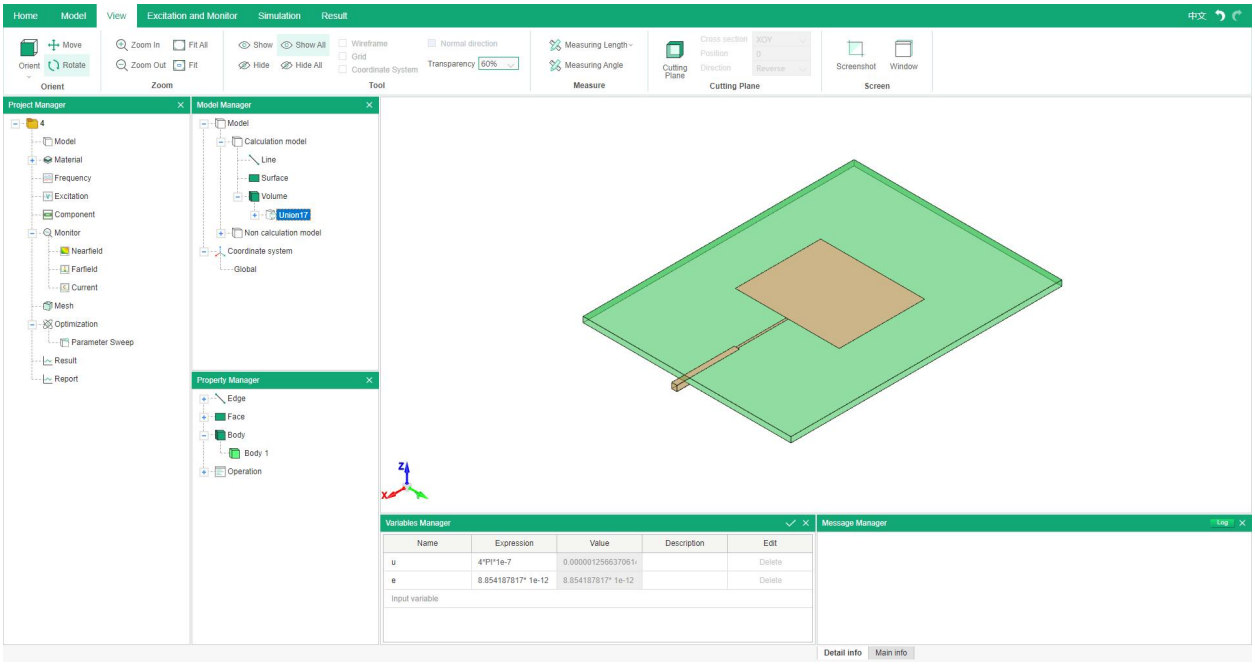


Figure 111 After selecting a transparency setting

4.3 Measure

The **Measure** function measures the geometric parameters of the modeling view interface. Measurement include **Measuring Length** and **Measuring Angle.**

4.3.1 Measure Length

The operation steps to measure the length are:

1、 Click **Measuring Length** in the **Measure** bar of the view area, and choose to measure the distance between two points.

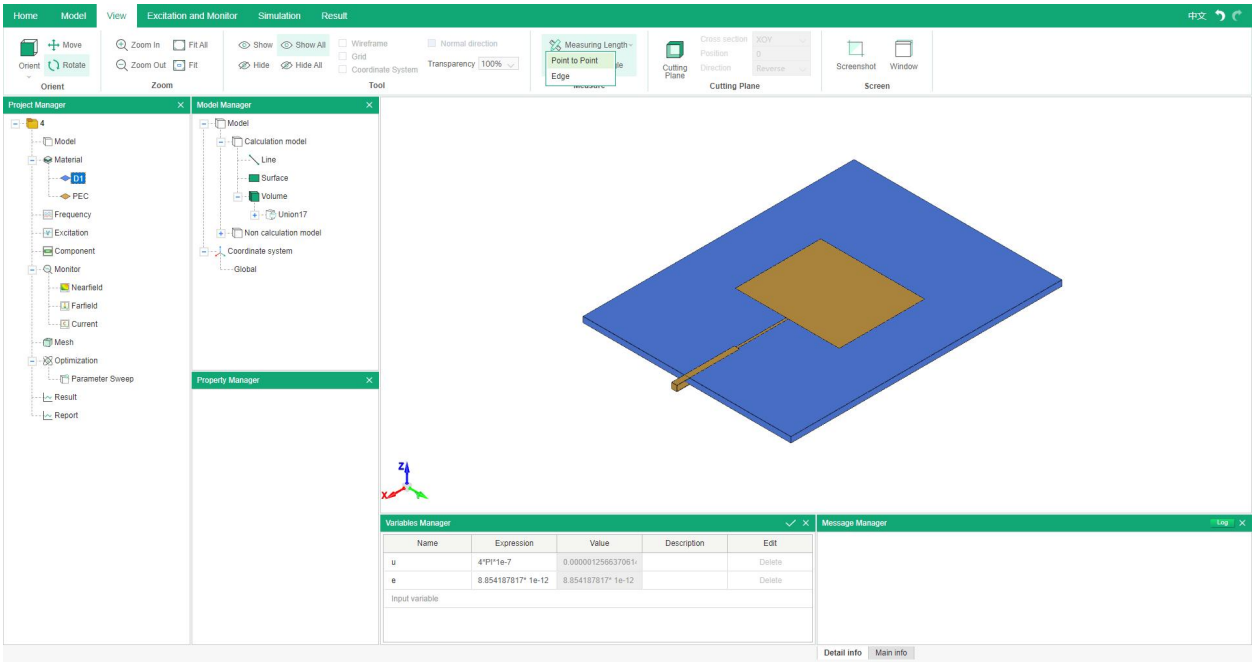


Figure 112 Measure length

2、 Double-click the left mouse button in the modeling view area, and select the starting point of the measurement length.

3、 Double-click the left mouse button in the modeling view area to select the final point of the measured length. At this time, the coordinates of the starting point, the coordinates of the ending point, the length between the starting point and the ending point, and the difference in x/y/z axis positions will be displayed in the red box below the view area.

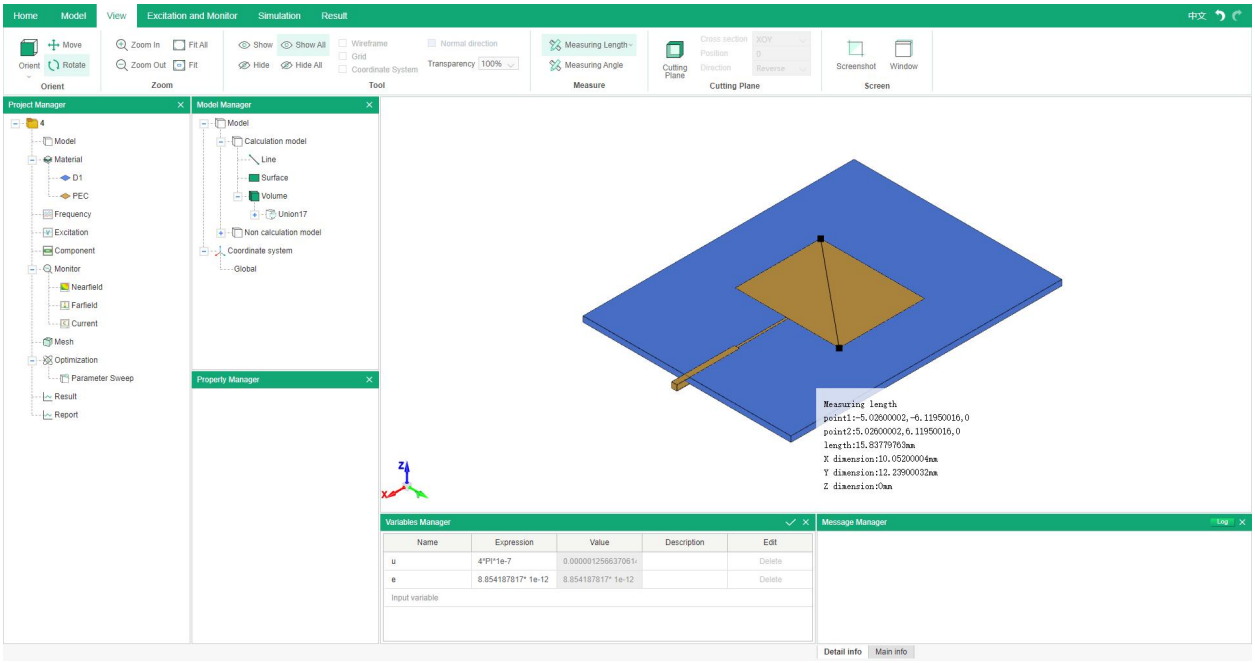


Figure 113 Measurement result

4、 Press the **Esc** button, the reference line for measuring length will disappear, and the measurement state will be canceled.

4.3.2 Measuring Angle

The steps of measuring angle are as follows:

1、 Click the **Measuring Angle** in the measurement bar of the view area.

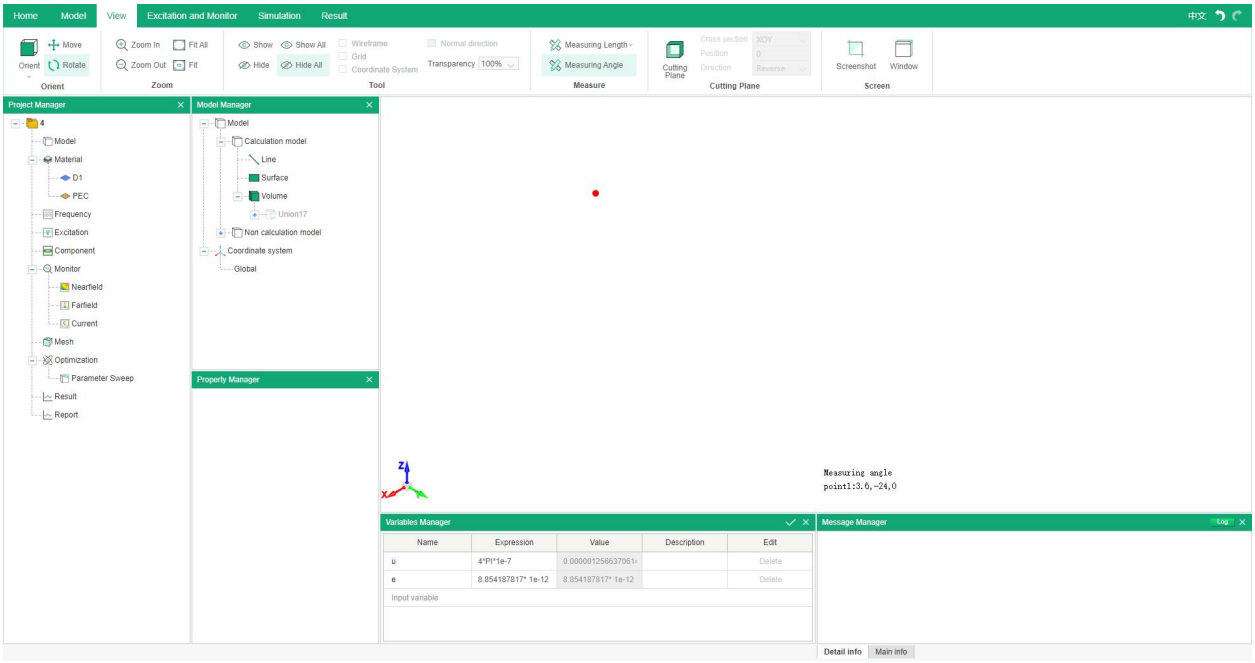


Figure 114 Measuring angle

2、 Double-click the left mouse button in the modeling view area, and select the starting point where the angle needs to be measured.

3、 Click the left mouse button in the modeling view area, and select the vertex that needs to measure the angle.

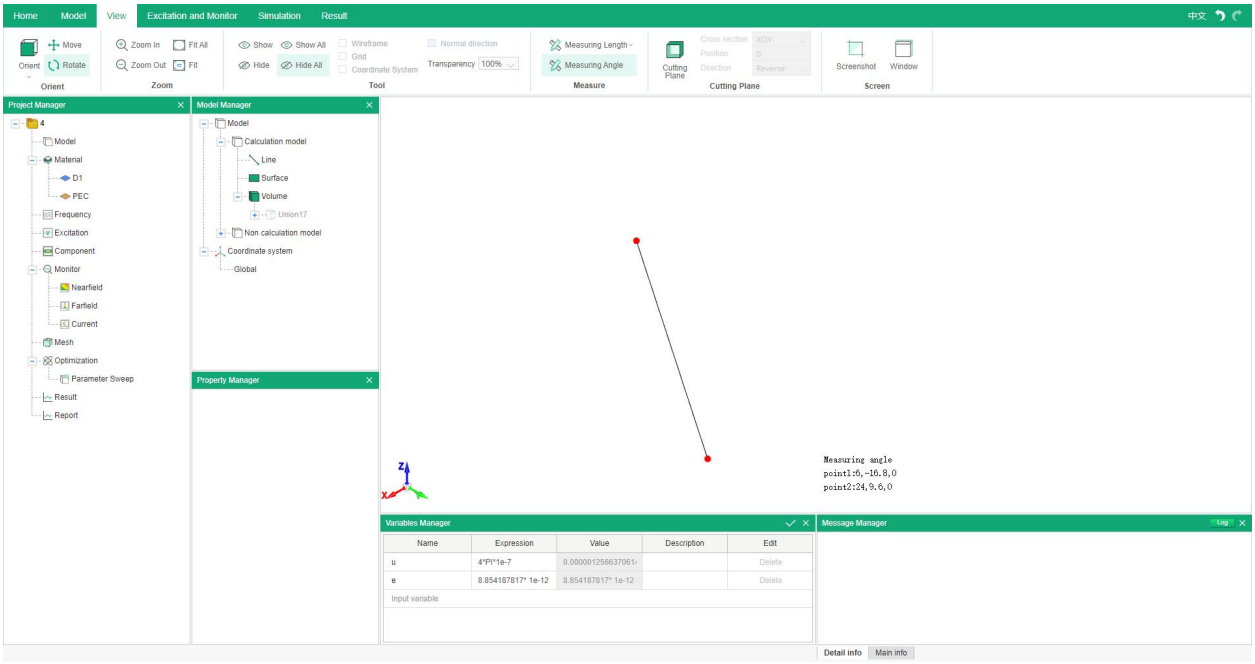


Figure 115 Choice of start side

4、 Finally, double-click the left mouse button in the modeling view area, and select the end point of the measurement angle. At this time, the coordinates of the starting point, the coordinates of the vertex, the coordinates of the end point, the starting point, the angle formed between the vertex and the end point will be displayed in the red box below the viewing area.

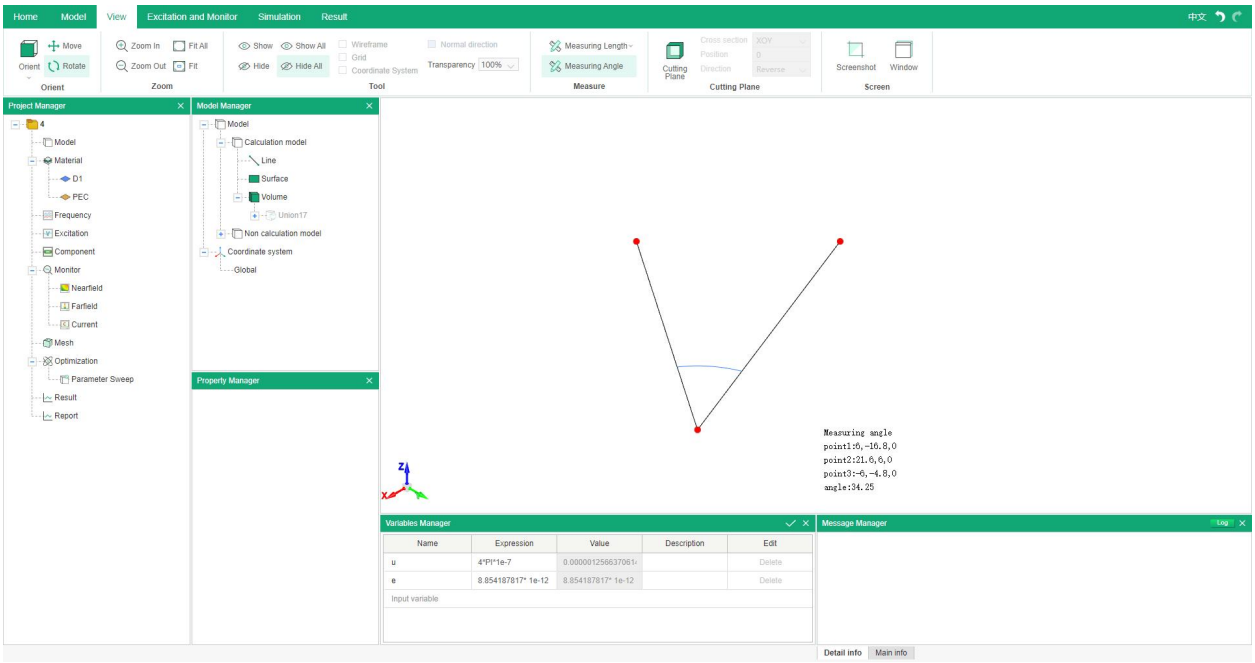


Figure 116 Measurement result

5、 Press the **Esc** button, the reference line for measuring the angle will disappear, and the measurement state will be canceled.

4.4 Cutting Plane

The **Cutting Plane** is used to view the model situation on both sides of the section intercepted by the cutting plane.

The steps to set the cutting plane are:

1、 Click **Cutting Plane** in the red frame of the section bar of the view area to add a section view.

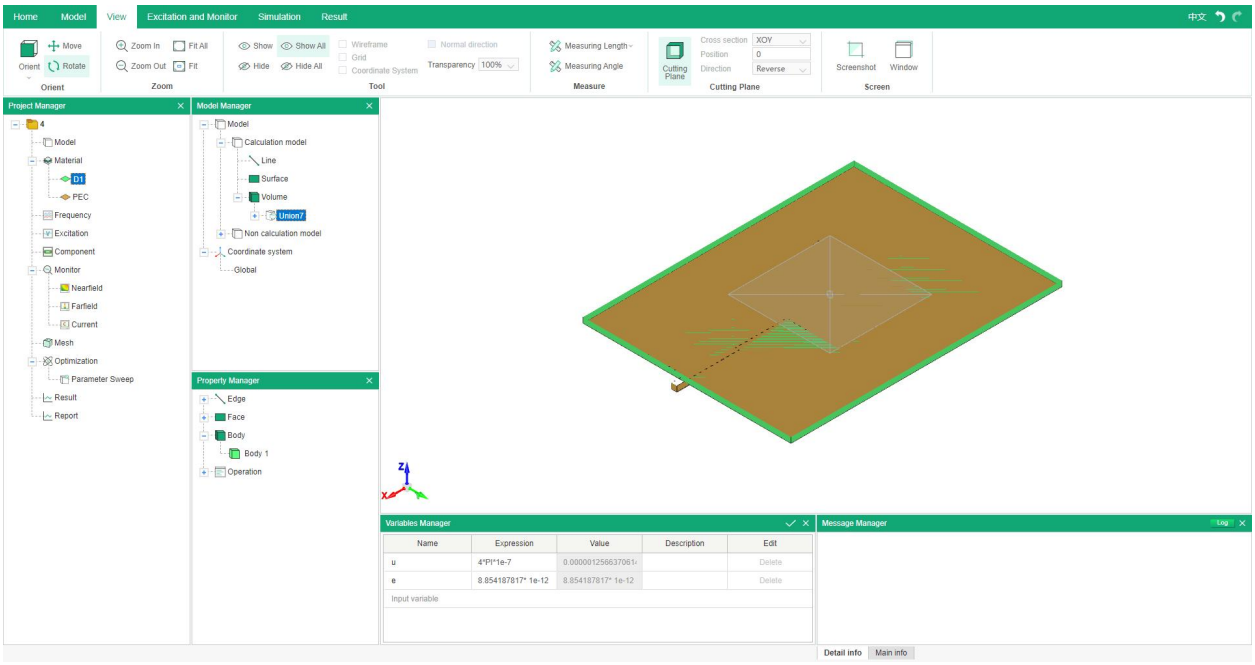


Figure 117 Click the section button

2、 Click the **Cross Section**, and select the direction of the section in the drop-down bar. You can choose the section to be located on the XOY plane, YOZ plane, ZOX plane or a free choice plane.

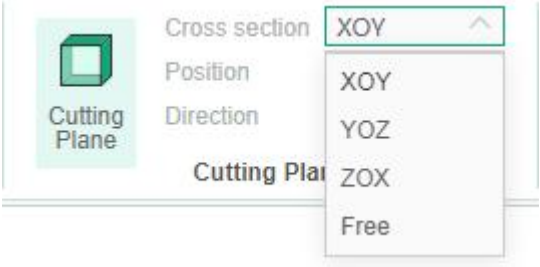


Figure 118 Cross section selection

3、 Click the **Position** to input the position coordinates of the cross section.

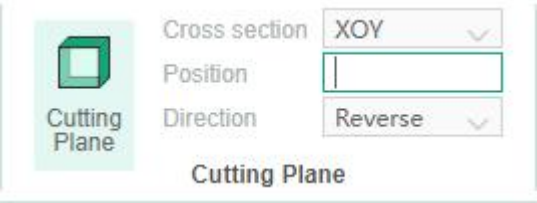


Figure 119 Location selection

4、 Click the **Direction** and select the section display direction in the drop-down bar, and you can choose the forward model display and reverse model display of the cross section.

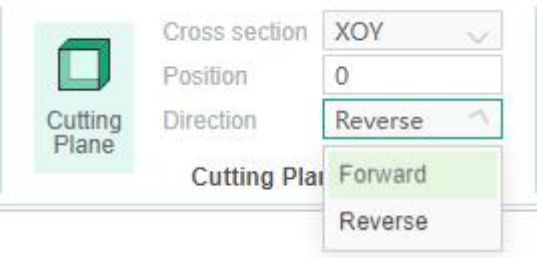


Figure 120 Direction selection

5、 Click the **Cutting Plane** again to cancel the display state of the section.

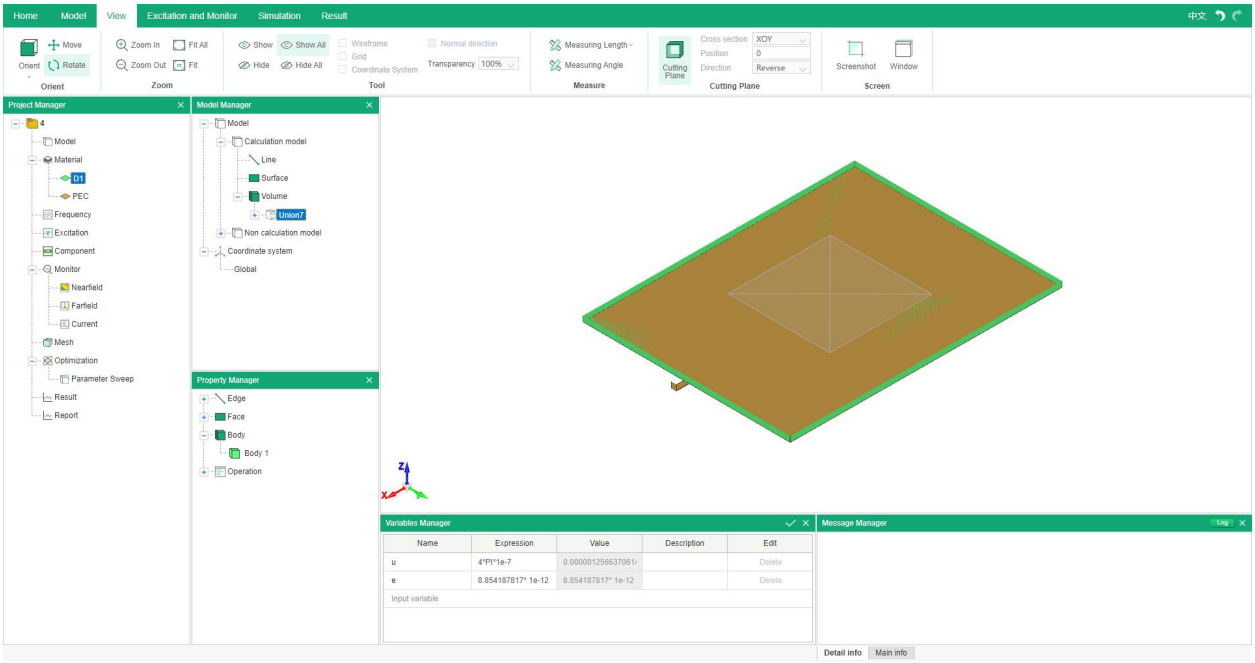


Figure 121 View interface after setting

4.5 Screen

4.5.1 Screenshot

The **Screenshot** captures all or part of the current interface and save it locally.

4.5.2 Window

Window chooses certain module bars to display. Click the window button, in the window selection, check the modules you want to display, you can choose display **Project Manager, Property Manager, Model Manager, Variables Manager, Message Manager**.

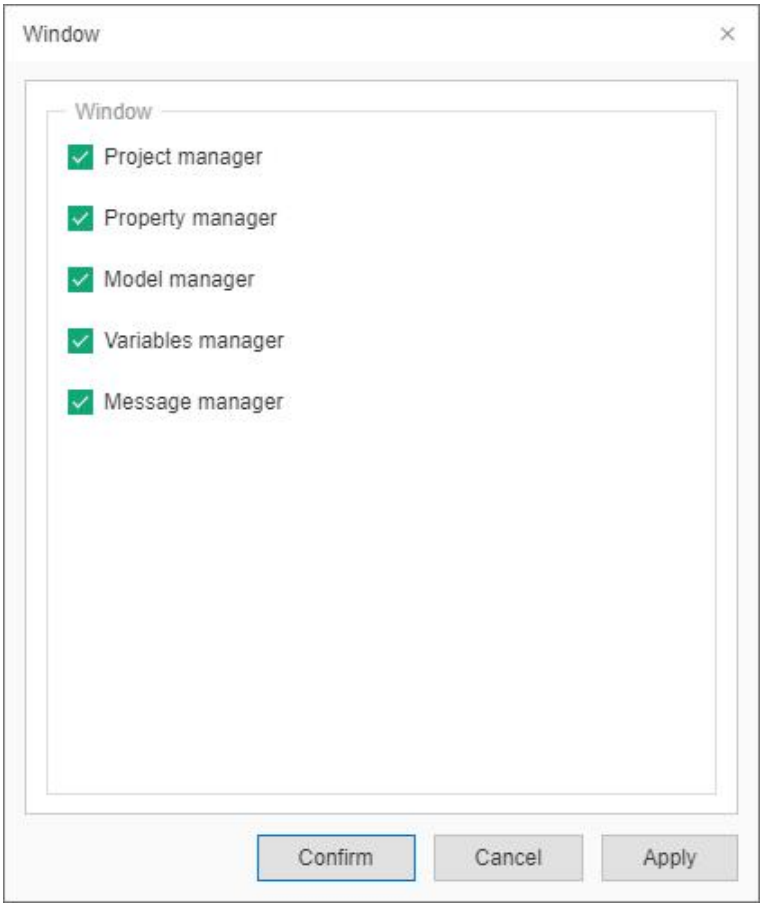


Figure 122 Window

5 Excitation and Monitor module

Excitation and Monitor module sets the simulation frequency, excitation source, synthesis source, and result settings. The excitation sources include **Connection Port, Waveguide Port, Lumped Port, Plane Wave, Electric Dipole, Magnetic Dipole, Near-Field Source, Far-Field Source** and **Spherical Wave**. Results settings include **Current** result setting, **Near Field** result setting and **Far Field** result setting.

5.1 Frequency

Frequency setting includes: **Single Frequency**, **Linear Interval**, **Log Interval** and **Discrete Point**. The frequency unit in frequency setting is the same as the one in unit setting.

5.1.1 Single frequency

The steps of frequency setting are:

1、 Click the **Frequency** button.

2、 Select a single frequency under the input box in the frequency window, and then enter frequency.

3、 Click **Confirm**.

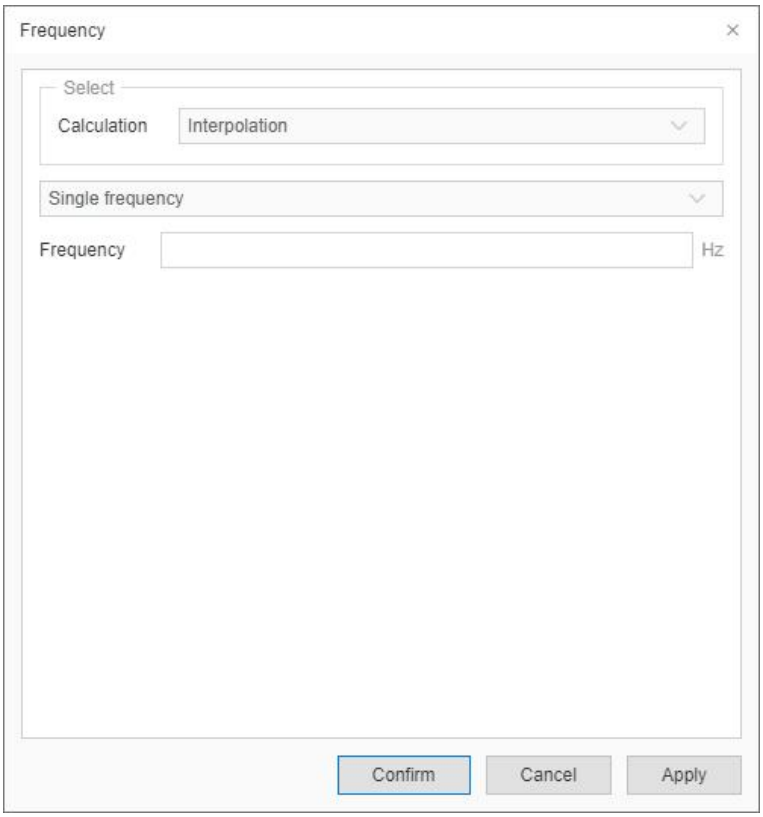


Figure 123 Single frequency point

5.1.2 Linear interval

The steps of frequency setting are:

1、 Click the **Frequency** button;

2、 Select Linear Interval, and input the start frequency, stop frequency and number of frequencies in sequence.

3、 Click **Confirm**.

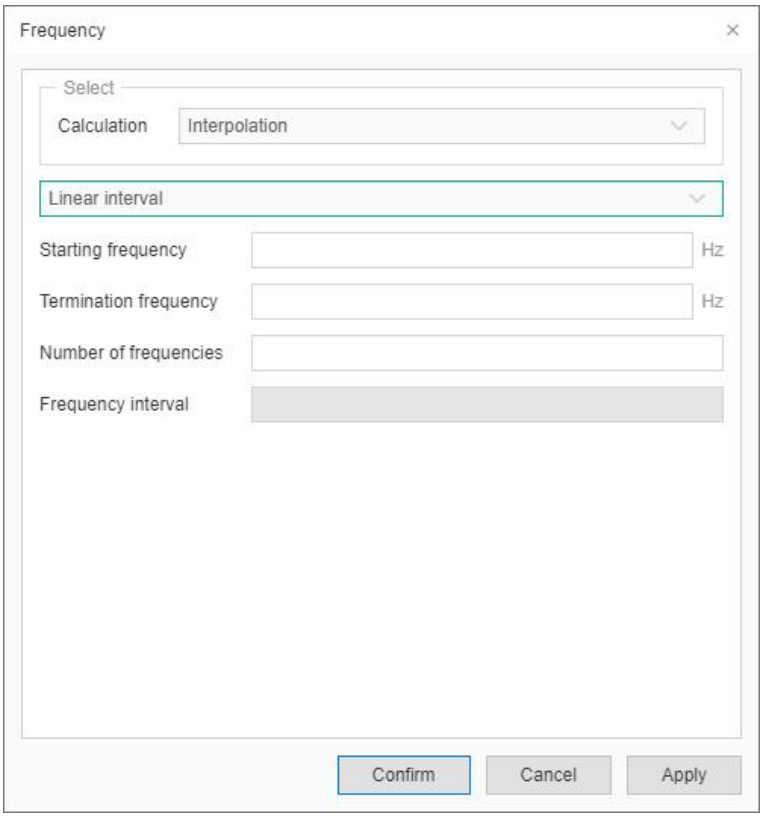


Figure 124 Linear interval

5.1.3 Log interval

The steps of frequency setting are:

1、 Click the **Frequency** button.

2、 Select log interval, and input the start frequency, stop frequency and number of frequencies in turn.

3、 Click **Confirm.**

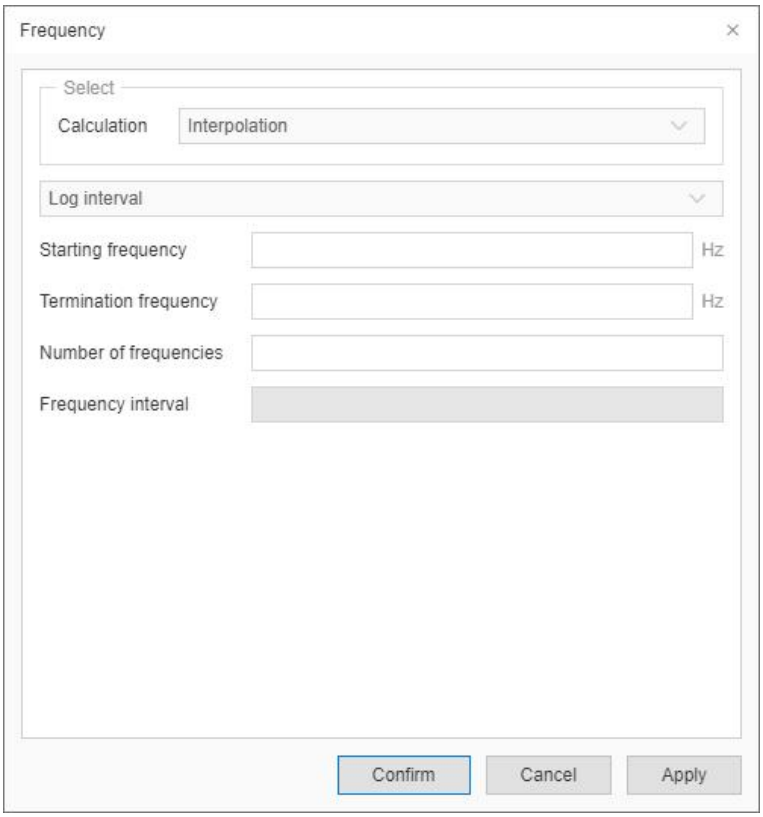


Figure 125 Log interval

5.1.4 Discrete point

The steps of frequency setting are:

1、 Click the **Frequency** button.

2、 Select discrete points, and then input discrete frequencies in sequence.

3、 Click the **Add** button to increase the number of input discrete points.

4、 Select the selection box before the frequency number and click **Delete** to delete the discrete point.

5、 Click **Import** to import discrete frequency points from the file.

6、 Click **Confirm**.

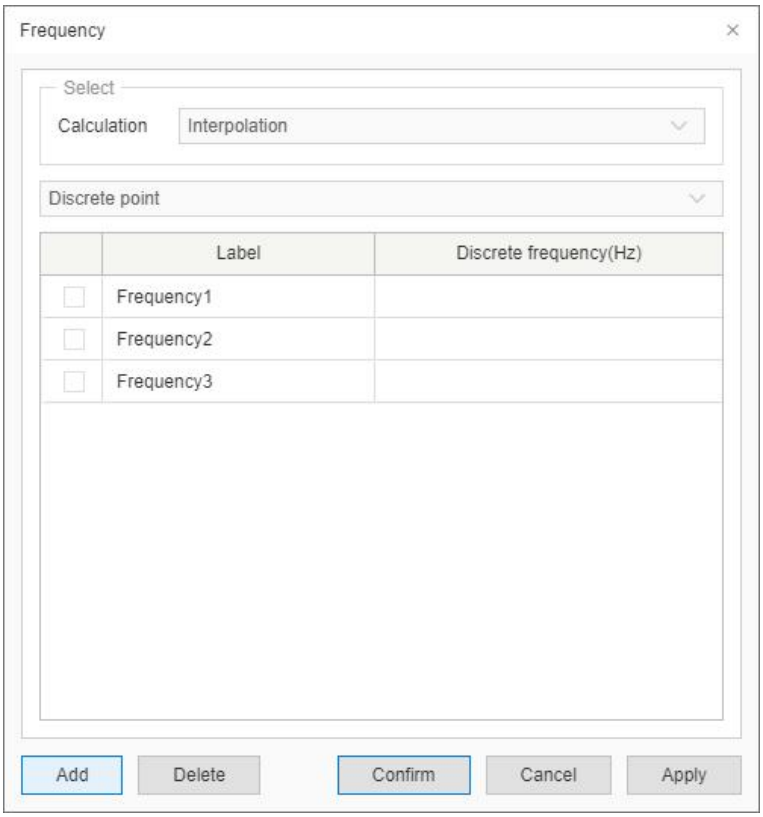


Figure 126 Discrete point

5.2 Port

Different simulation models require different excitation. The excitation settings include ports (connection port, waveguide port, and lumped port) and ideal sources (plane wave, electric dipole, and magnetic dipole), Near-field source, far-field source and spherical wave. The excitation synthesis function realizes the synthesis between individual sources.

5.2.1 Connection port

Connection port (voltage source excitation). In the port setting, enter the line name (or click on the target line), port label name, port position, excitation voltage amplitude, phase and internal resistance value.

The operation steps of creating a connection port are as follows:

1、 Click **Connection port**.

2、 In the pop-up window, select the line model in the view area or enter the line name.

3、 Enter the port name (the default value is connection port 1, 2...).

4、 Select the port position under the position box, if select other, enter the port position in the input box after the position (0-100, where 0 represents the starting point, 100 represents the end point.

5、 In the excitation input box, input the real part and imaginary part of voltage, phase and internal resistance in sequence (the default values: voltage is 1, phase is 0, real part of internal resistance is 50, and imaginary part is 0).

6、 Click **Confirm**.

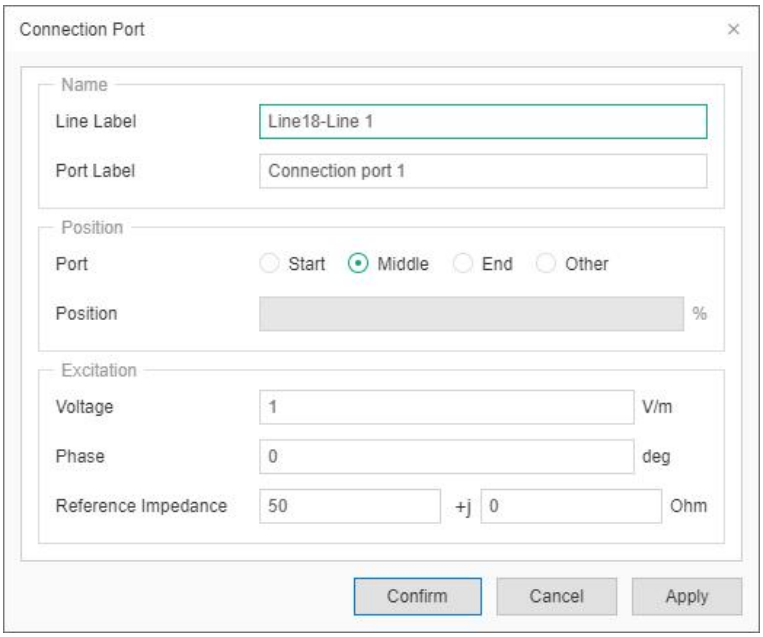


Figure 127 Connection port

7、 The connection port will appear on the line model in the view area.

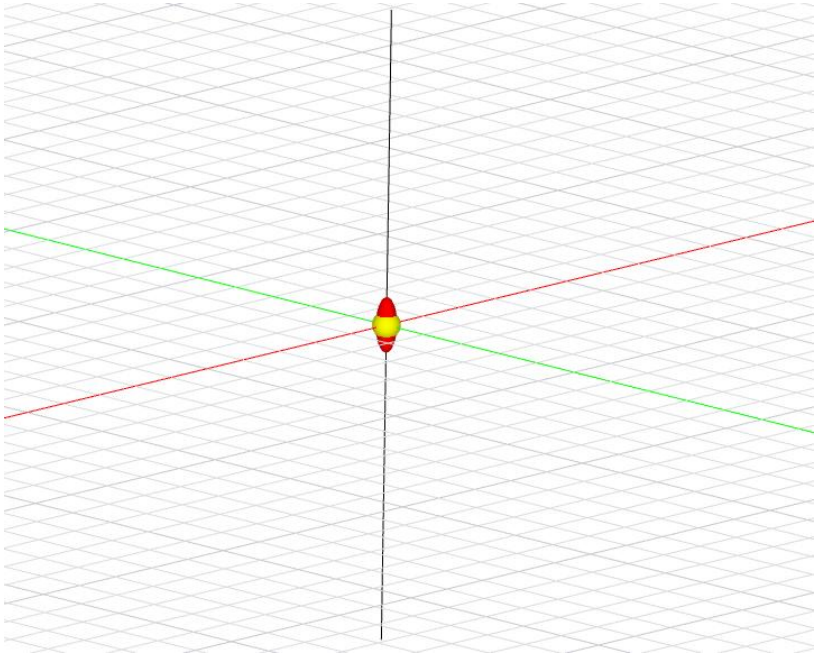


Figure 128 Connection port

8、 A connection port label name will appear in the sub-node activated in the project manager area. Double-click the label name, the connection port property window will pop up to be modified.

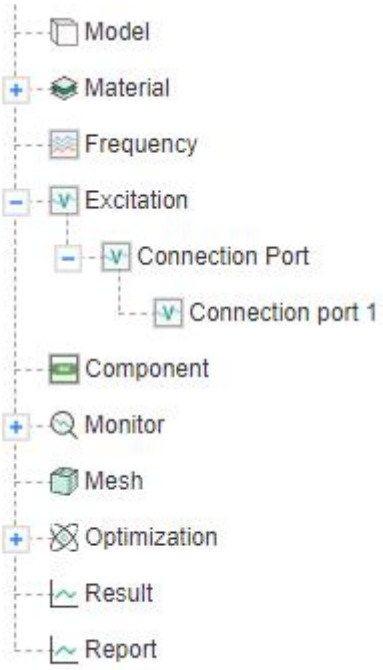


Figure 129 Connection port label name

5.2.2 Waveguide port

Waveguide port is generally applicable for waveguide structures. In the waveguide port setting, enter the surface name, port name, wave port propagation direction and wave mode of the wave port setting.

1、 Method 1

The steps of waveguide port setting are as follows:

(1) Click the **Waveguide Port.**

(2) In the waveguide port window, select the surface to be set as a wave port in the model view or enter the surface name.

(3) Select the propagation direction of the port in the Propagation Direction box.

(4) In the Mode box, choose Main Mode or Custom Mode :

(5) If the main mode is selected, the number of modes is 1. Select the direction of the integration line, characteristic impedance, and enter amplitude and phase in the box below.

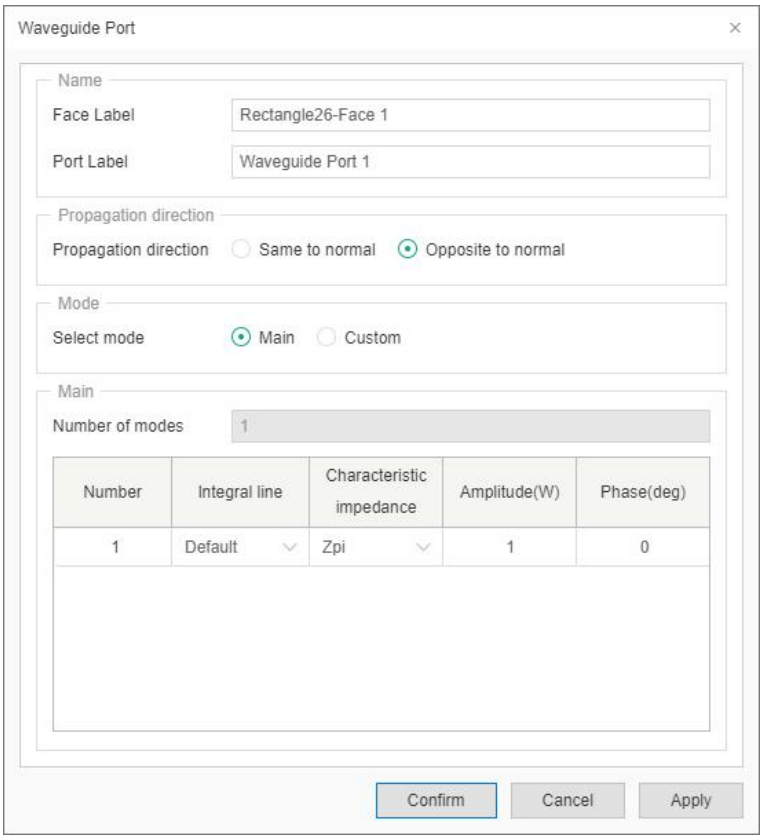


Figure 130 Waveguide port - main mode

If custom mode is chosen, enter the number of modes and select the direction of the integration line, the characteristic impedance, and the input amplitude and phase in each mode in the box below.

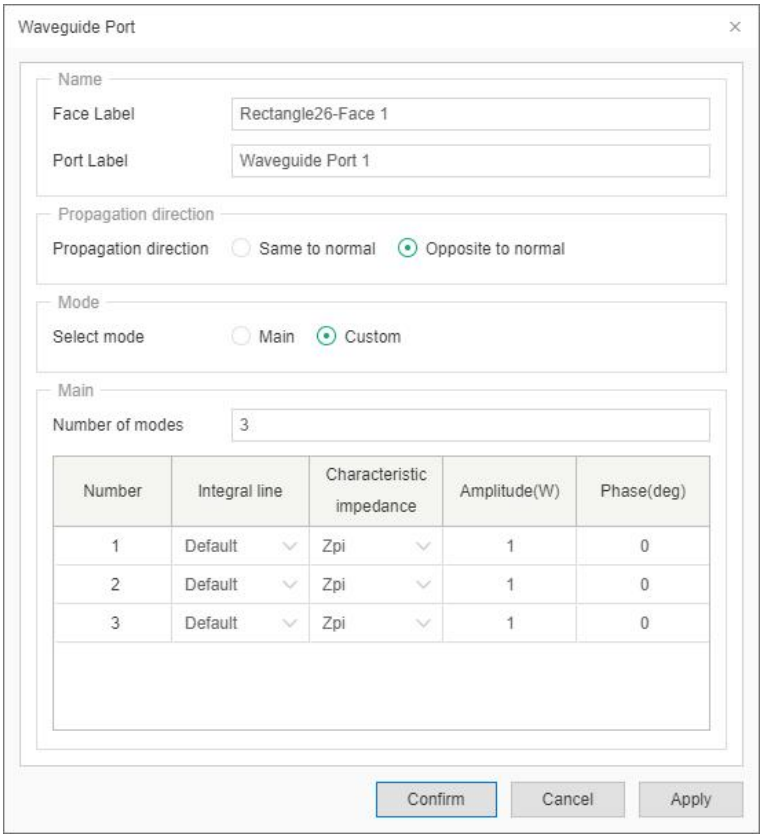


Figure 131 Waveguide port - custom mode

(6) Click **Confirm** to complete waveguide port settings.

(7) After completing the settings, a waveguide port will appear in the view area.

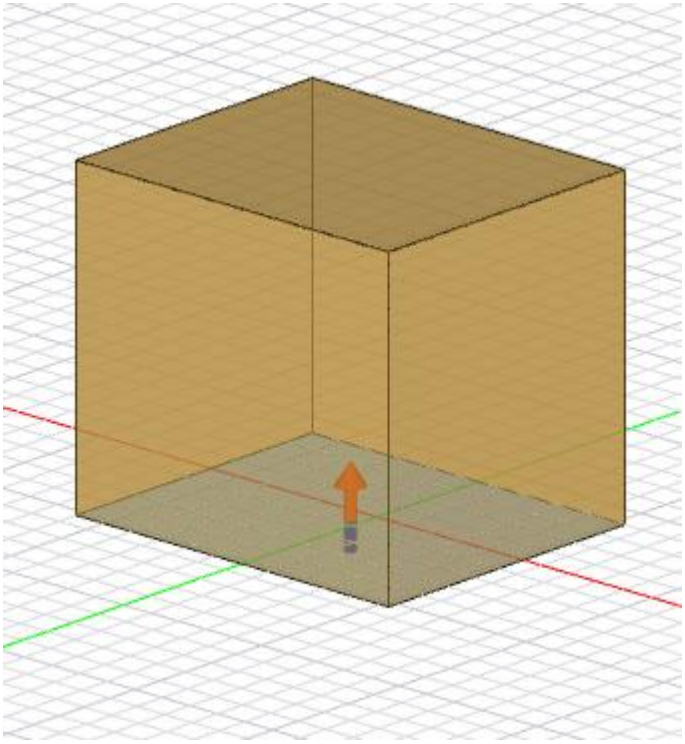


Figure 132 Waveguide port

(8) Waveguide port label name will appear in the sub-node of the excitation in the project manager area. Click on the label name, and the waveguide port property window will pop up to be modified.

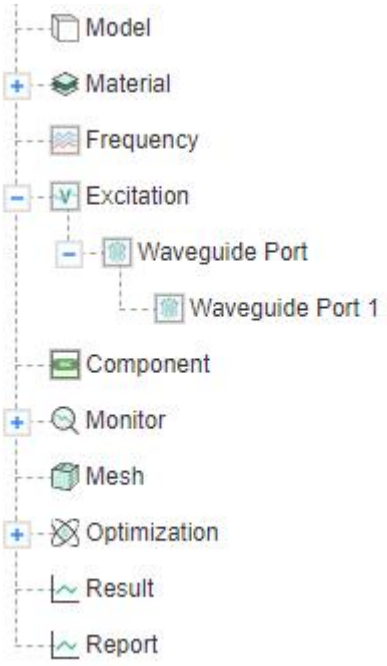


Figure 133 Waveguide port label name

2、 Method 2

The steps of waveguide port setting are as follows:

(1) Select the model that needs to set waveguide port in the model manager area, and then select the surface label to set the wave port in the property manager area - surface model.

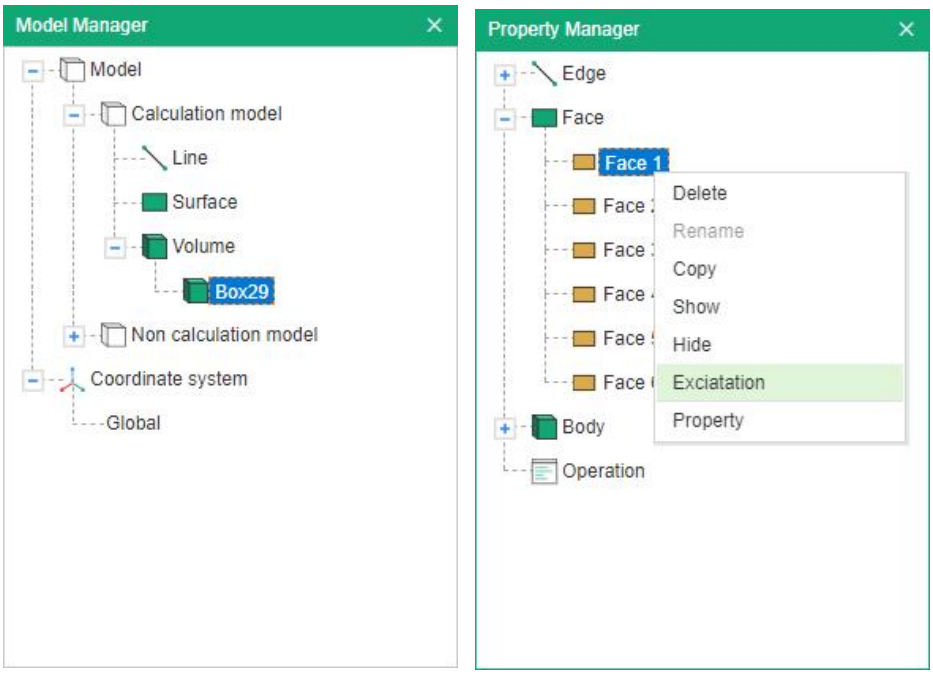


Figure 134 Waveguide Port - selects the face

(2) Right-click and select Excitation.

(3) Select the propagation direction of the waveguide port in the Propagation Direction box.

(4) In the Mode box, choose Main Mode or Custom Mode :

If the main mode is selected, the number of modes is 1. Select the direction of the integration line, characteristic impedance, and enter amplitude and phase in the box below.

If custom mode is chosen, enter the number of modes and select the direction of the integration line, the characteristic impedance, and the input amplitude and phase in each mode in the box below.

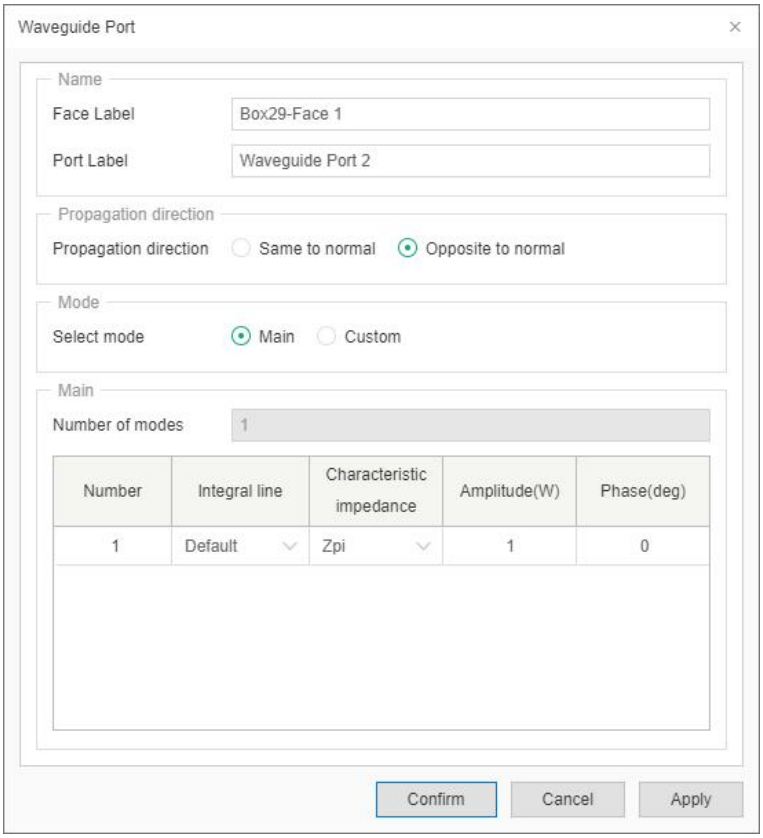


Figure 135 Waveguide port - main mode

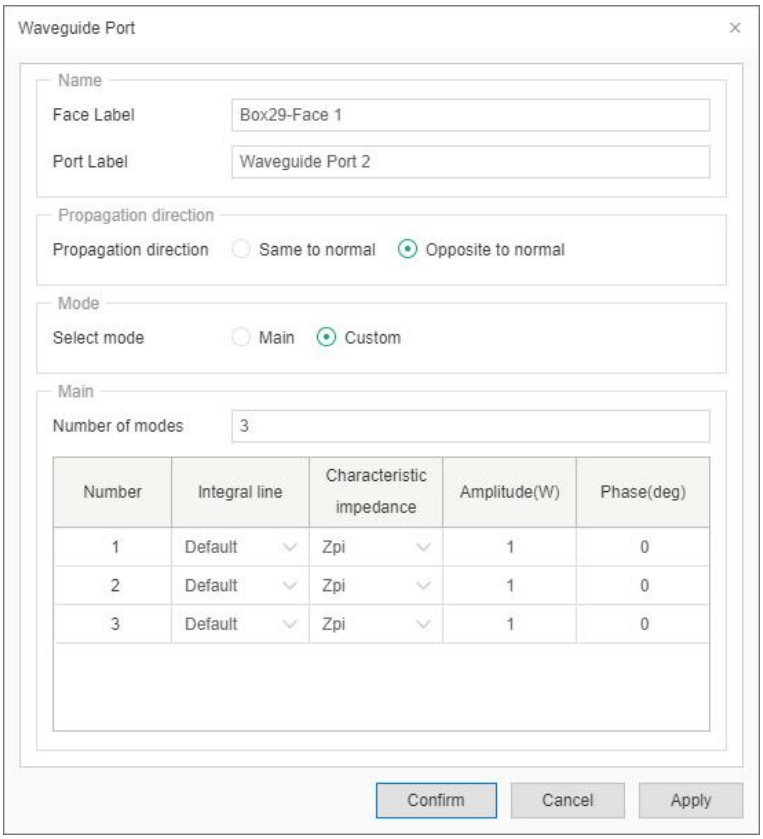


Figure 136 Waveguide port - custom mode

(5) Click **Confirm** to finish.

(6) A waveguide port will appear in the view area.

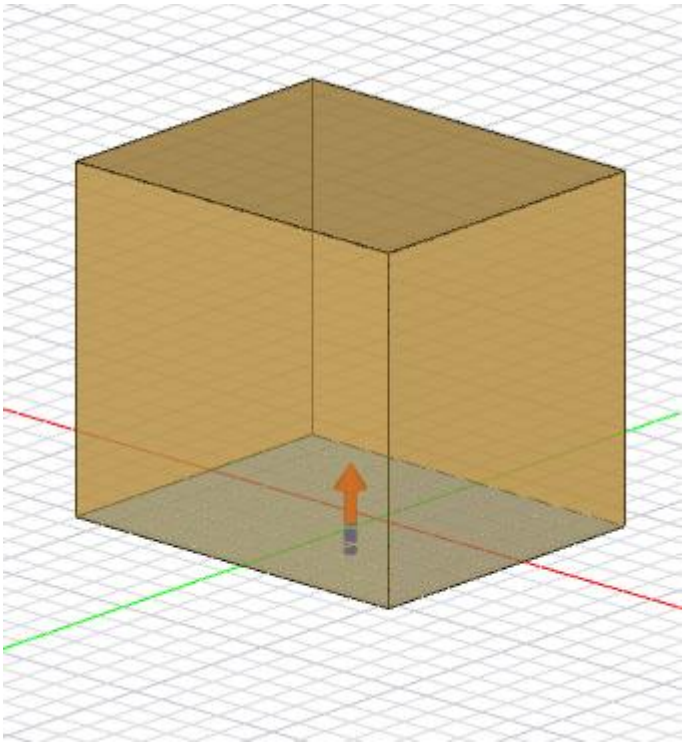


Figure 137 Waveguide port

(7) Waveguide port label name will appear in the sub-node of the excitation in the project manager area. Click on the label name, and the waveguide port property window will pop up to be modified.

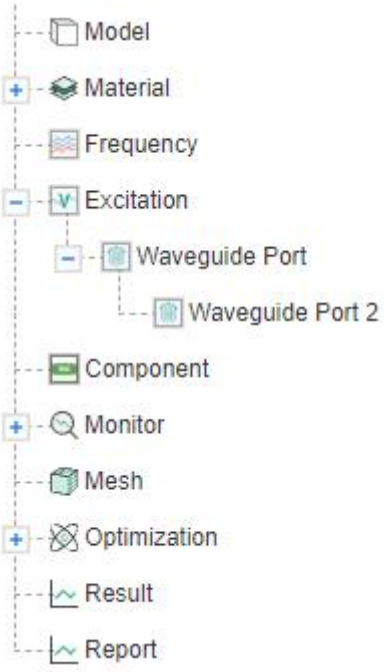


Figure 138 Waveguide port label name

5.2.3 Lumped port

Lumped port are usually applied for transmission line models. In the lumped port setting, select the positive and negative sides of the lumped port, port name, excitation mode, amplitude, phase and internal resistance.

The steps of lumped port excitation setting are:

1、 Click **Lumped port**.

2、 When lumped port window pops up, select the positive side and the negative side of the lumped port in the view interface to determine the port position.

3、 In excitation mode, selects voltage source or current source:

If voltage source is selected, input the amplitude (default is 1), phase (default is 0), real part of internal resistance (default is 50) and imaginary part (default is 0) in sequence.

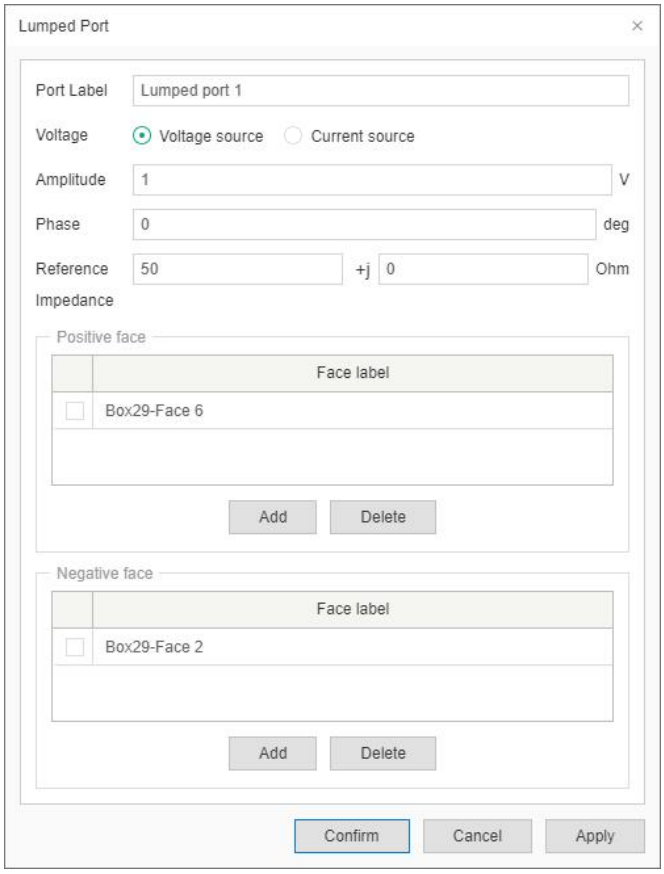


Figure 139 Lumped port - voltage source

If current source is selected, input the amplitude (default is 1), phase (default is 0), real part of internal resistance (default is 50) and imaginary part (default is 0) in sequence.

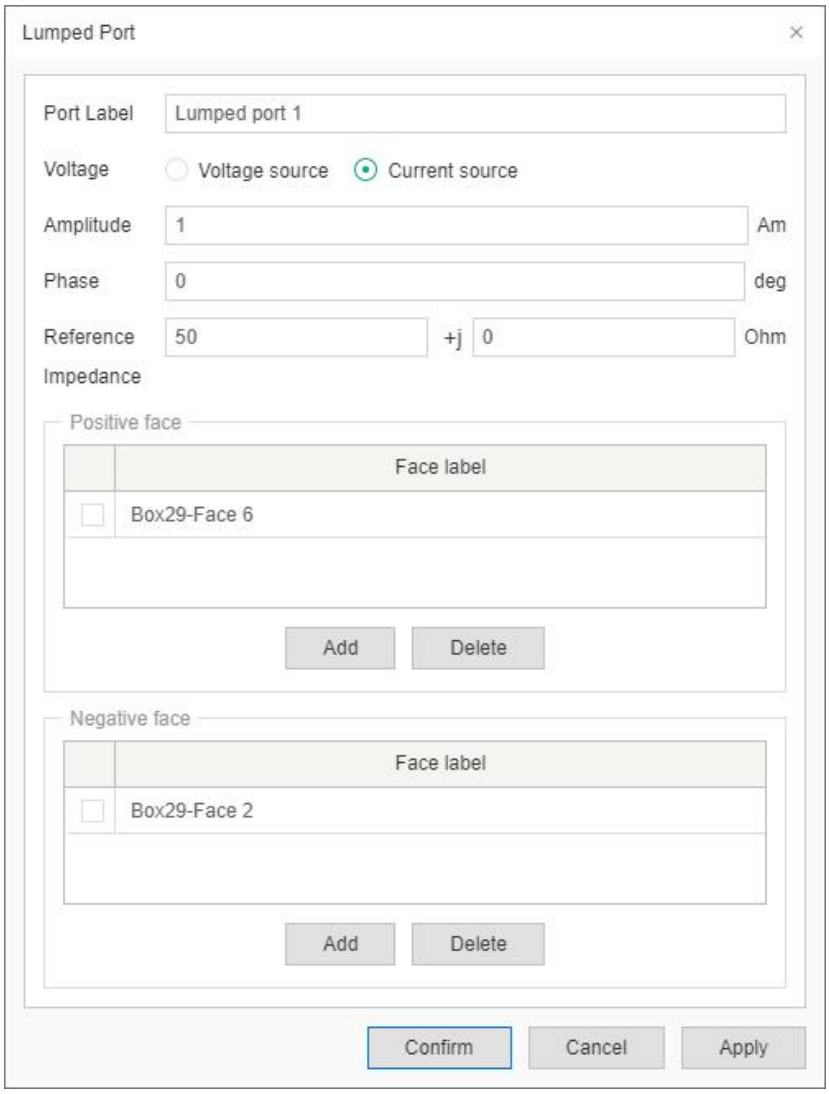


Figure 140 Lumped port - current source

4、 Click **Confirm** to finish.

5、 A lumped port will appear in the view area.

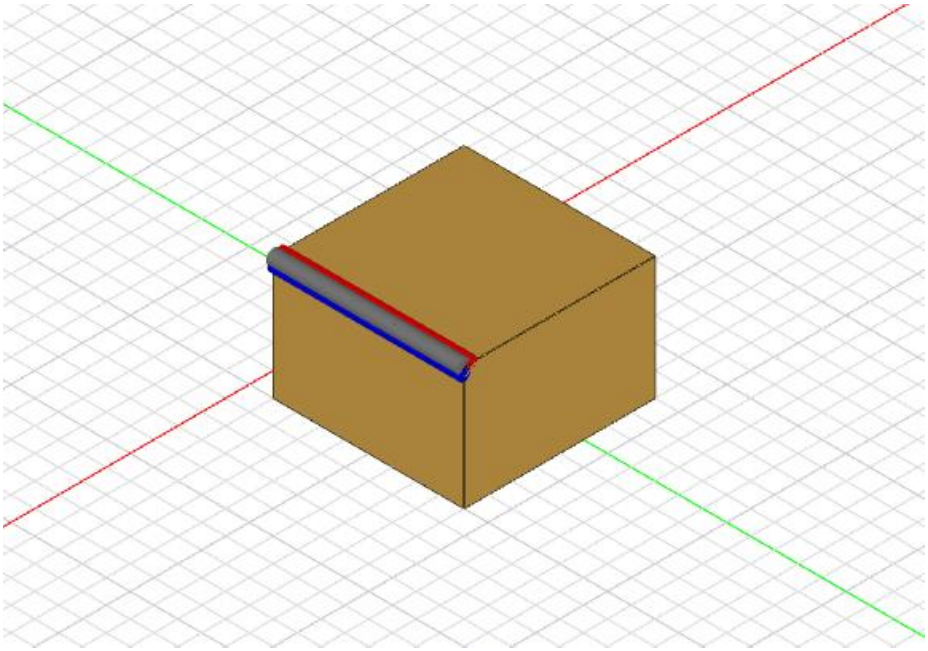


Figure 141 Lumped port

6、 The lumped port label name will appear in the sub-node activated in the project manager area. Click on the label name a lumped port window will pop up to be modified.

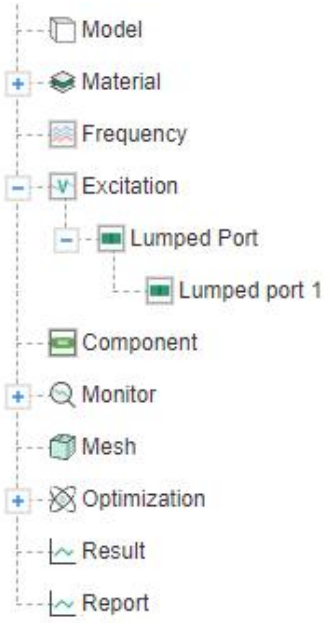


Figure 142 Lumped port label name

5.3 Ideal source

5.3.1 Plane wave

Plane wave can be applied to any structure, mainly for scattering simulations, especially for RCS. In the plane wave setting, input the label name, select a single incident wave or circular incident waves in multiple directions. In the plane wave incident direction column, enter the start point, end point, interval of theta and the start point, end point, interval of phi in sequence. Finally, in polarization, enter the amplitude and phase of theta polarization and phi polarization accordingly.

The steps of plane wave excitation setting are as follows:

1、 Click **Plane Wave**.

2、 In the plane wave setting window, enter the label name of the plane wave in the label name (the default is plane wave + number).

3、 Select a single incident wave or cycle in multiple directions.

i. If a single incident wave is selected, in the incident direction box, input the starting point of theta and phi.

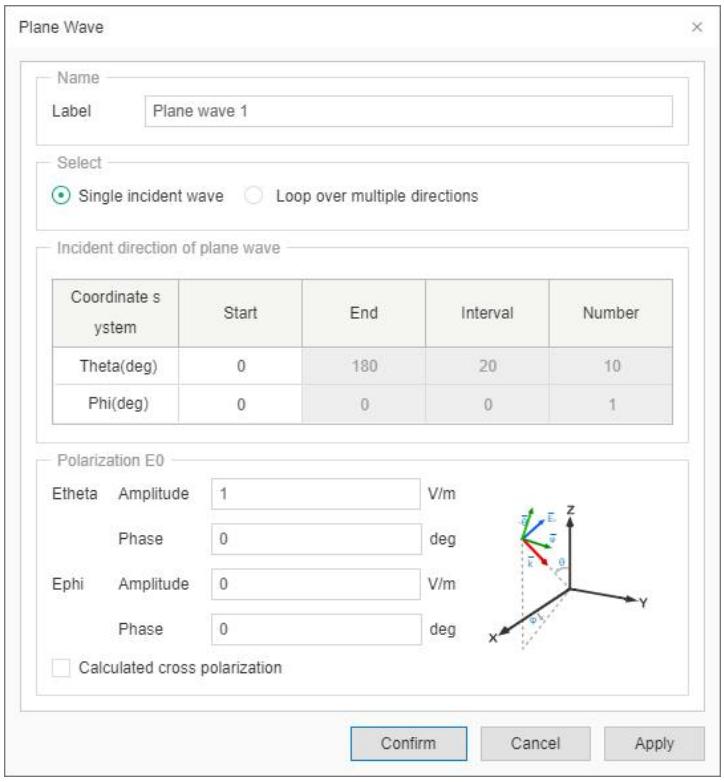


Figure 143 Plane wave - single incident wave

ii. If to cycle in multiple directions is selected, in the incident direction box, enter the start point, end point, and interval of theta and phi (the number will be calculated automatically).

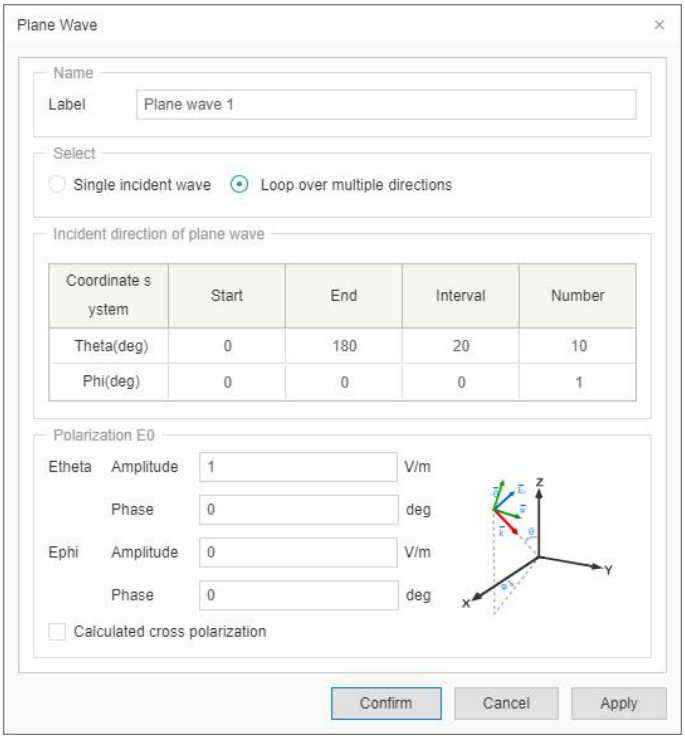


Figure 144 Plane wave - loops in multiple directions

4、 In the polarization box, enter the amplitude and phase of theta polarization and phi polarization in sequence.

5、 Choose whether to check the calculation of cross polarization

6、 Click **Confirm** to finish.

7、 A plane wave will appear on the wire model in the viewing area.

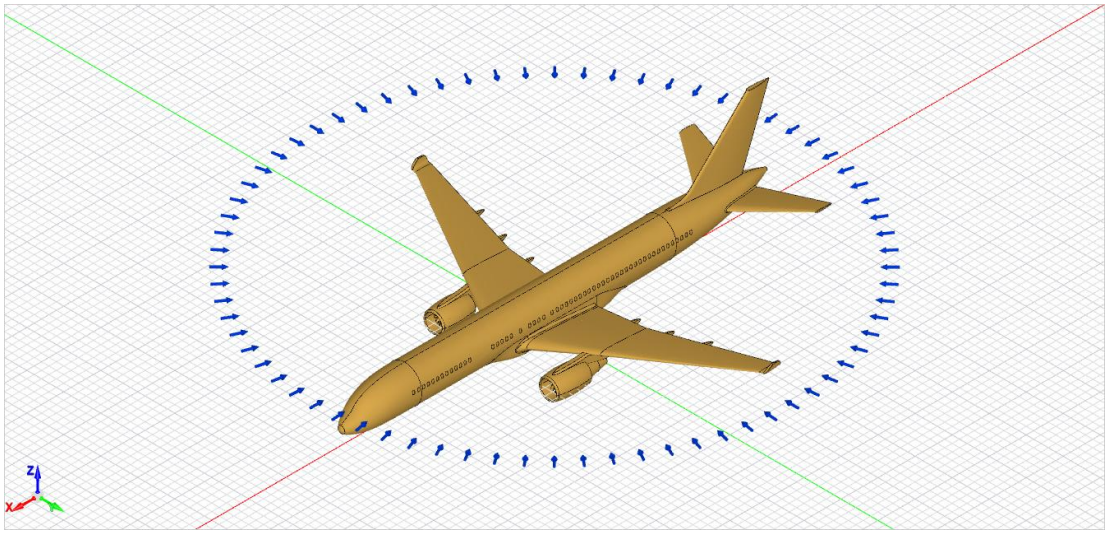


Figure 145 Plane wave

8、 The plane wave-plane wave label name will appear in the sub-node of the excitation in the project manager area. Click the label name, and window for plane wave settings will pop up to be modified.

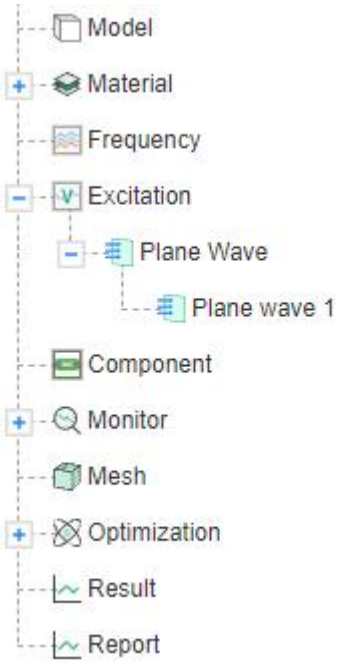


Figure 146 Plane wave tag name

Note:

If to cycle in multiple directions, the starting point and ending point of theta or phi can be the same, and the interval input should be 0.

In polarization E0, if the amplitude and phase of phi are both 0, it means that the incident wave is theta polarized. if the amplitude and phase of theta are both 0, it means that the incident wave is phi polarized. if the amplitude and phase of theta and phi are If both are 0, the polarization mode is the vector sum of theta and phi angles.

5.3.2 Electric dipole

Electric dipole excitation is can be applied for any electromagnetic structure. In the electric dipole setting, enter the label name, input the amplitude and phase in the excitation column, and input the position and orientation of the electric dipole in the position and orientation column.

The steps of electric dipole excitation setting are as follows:

1、 Click **Electric Dipole**.

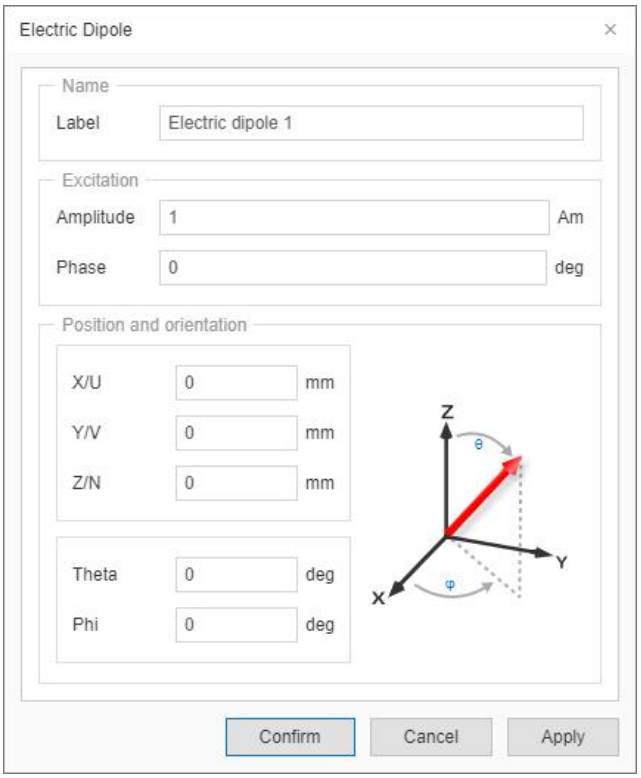


Figure 147 Electric dipole

2、 In the electric dipole setting window, enter the label name of the electric dipole in the name box (the default is Electric dipole + number).

3、 In the excitation box, enter the amplitude and phase in turn.

4、 In the position and orientation box, input the position in the X/U axis, Y/V axis, Z/N axis direction and the orientation in the Theta and Phi directions added by the electric dipole in sequence.

5、 Click **Confirm** to finish.

6、 An electric dipole will appear in the viewing area.

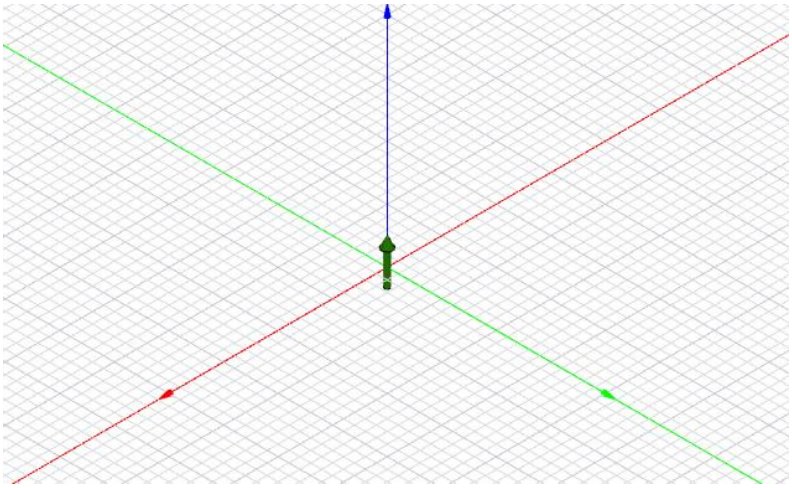


Figure 148 Electric dipole

7、 **Electric Dipole** label name will appear in the sub-node of the excitation in the project manager area. Click the label name, and the electric dipole window will pop up to be modified.

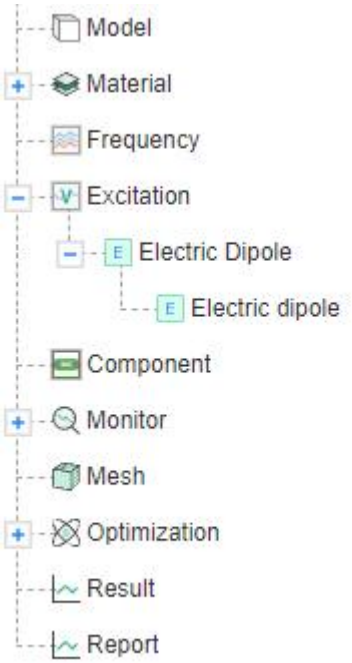


Figure 149 Electric dipole label name

5.3.3 Magnetic dipole

Magnetic dipole excitation is applicable for any electromagnetic structure. In the magnetic dipole setting, enter the label name, input the amplitude and phase in the excitation column, and input the position and orientation of the magnetic dipole in the position and orientation column.

The steps of magnetic dipole excitation setting are as follows:

1、 Click **Magnetic Dipole**.

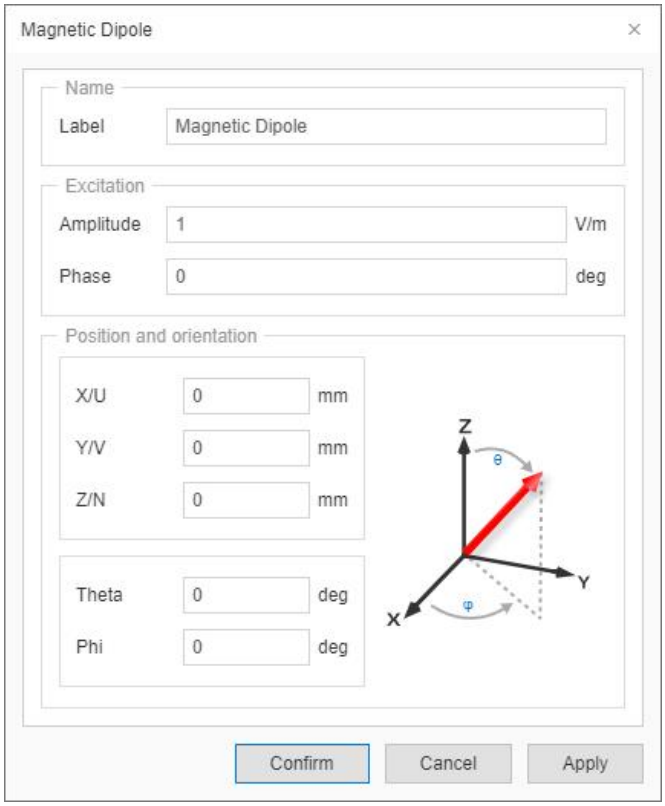


Figure 150 Magnetic dipole

2、 In the magnetic dipole setting window, first enter the label name of the magnetic dipole in the name box (the default is Magnetic Dipole + number).

3、 In the excitation box, enter the amplitude and phase in turn.

4、 In the position and orientation box, input the position in the X/U axis, Y/V axis, Z/N axis direction and the orientation in the Theta and Phi directions added by the magnetic dipole in sequence.

5、 Click **Confirm** to finish.

6、 A magnetic dipole will appear in the viewing area.

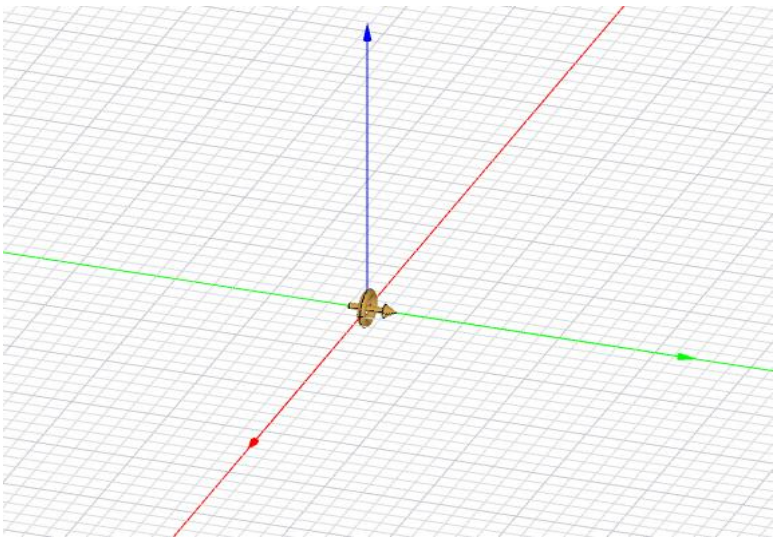


Figure 151 Magnetic dipole

7、 **Magnetic Dipole** label name will appear in the sub-node of the excitation in the project manager area. Click the label name, and the magnetic dipole bullet window will pop up to be modified.

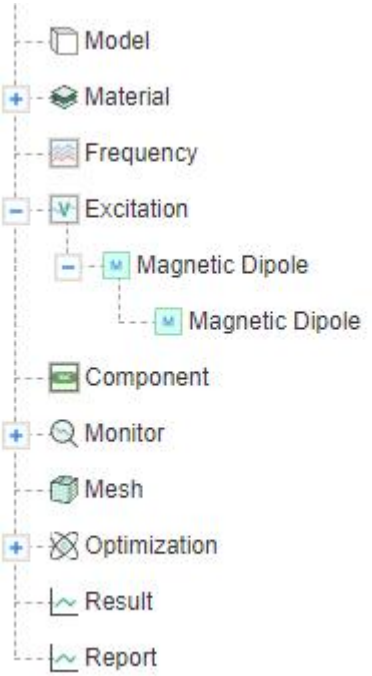


Figure 152 Magnetic dipole label name

5.4 Field source

5.4.1 Near Field

In the near-field source setting, enter the label name, amplitude and phase in sequence in the

excitation column, select the format of the near-field source in the field format drop-down box, and import the local near-field source file for simulation.

The steps of near-field source excitation setting are as follows:

1、 Click **Near-Field**.

2、 In the near -field source setting window, enter the label name of the near-field source in the name box (the default is Near Field + number).

3、 Enter the magnitude and phase of the excitation in Excitation.

4、 Select the format to add near-field sources in the field format, which is divided into electric field, magnetic field, and electromagnetic field settings.

5、 Import near -field source file (.enf).

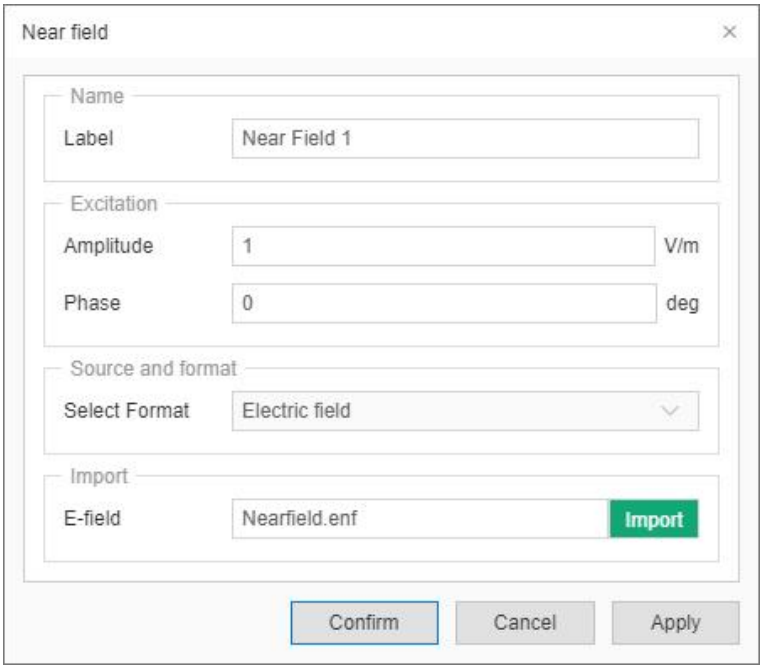


Figure 153 Near -field source

6、 Click **Confirm** to finish.

7、 A near-field source will appear in the viewing area.

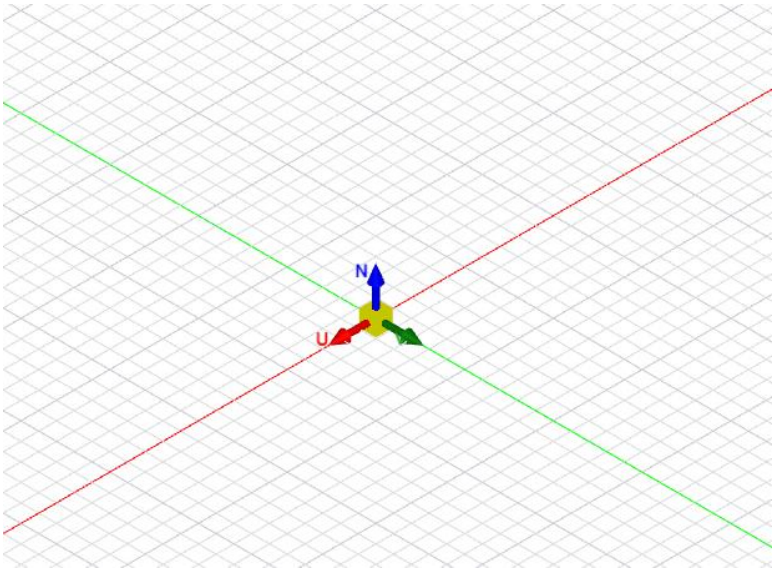


Figure 154 Near-field source

8、 The near-field source label name will appear in the sub-node of the excitation in the project manager area. Click the label name, and the window for near -field source settings will pop up to be modified.

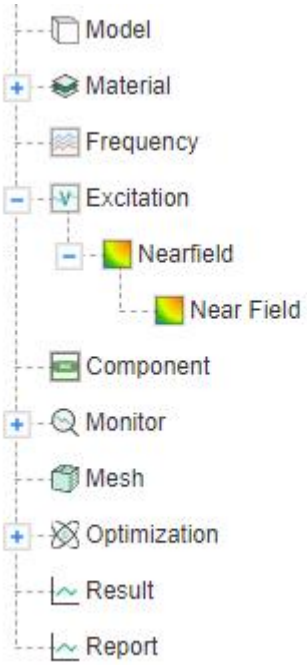


Figure 155 Near-Field source label name

5.4.2 Far Field

In the far-field source setting, enter the label name, input the amplitude and phase in the excitation column, input the position and orientation of the far-field source in the position and orientation column, and import the local near-field source file for simulation.

The steps of far-field source excitation setting are as follows:

1、 Click **Far-Field**.

2、 In the far field source setting window, enter the label name of the far field source in the name box (the default is Far field + number).

3、 Enter the magnitude and phase of the excitation in Excitation.

4、 Select the import file type in the import box, which is divided into far-field files and far-field CAE files, and then select the far-field files to be imported (.ff/.ffp).

8、 In the position and orientation box, input the position in the X/U axis, Y/V axis, Z/N axis direction and the orientation in the Theta and Phi directions added by the electric dipole in sequence.

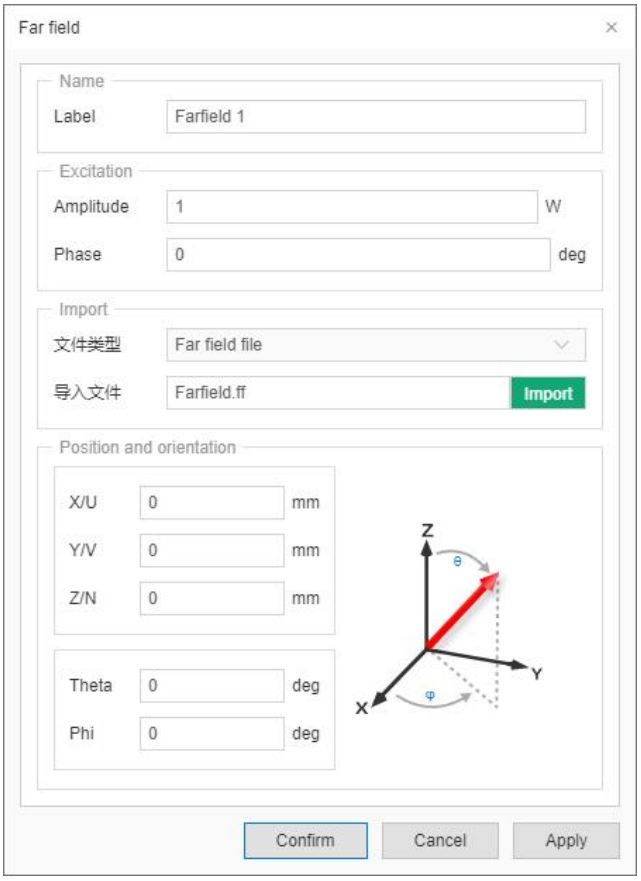


Figure 156 Far field source

5、 Click **Confirm** to finish.

6、 A far-field source will appear in the viewing area.

7、 The far field - far field source label name will appear in the sub-node of the excitation in the project manager area. Click the label name, and a far field source window will pop up to be modified.

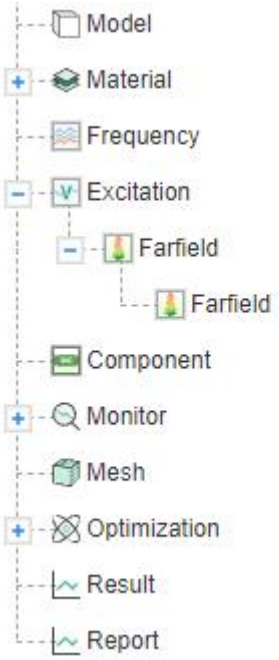


Figure 157 Far field source label name

5.4.3 Spherical Wave

In the spherical wave setting, the user needs to enter the label name, input the amplitude and phase in the excitation column, input the position and orientation of the spherical wave, and import the local ground wave file for simulation.

The steps of spherical wave excitation setting are as follows:

1、 Click **Spherical Wave**.

2、 In the spherical wave setting window, enter the label name of the spherical wave in the name box (the default is Spherical wave + number).

3、 Enter the magnitude and phase of the excitation in Excitation.

4、 Import spherical wave file (.ff).

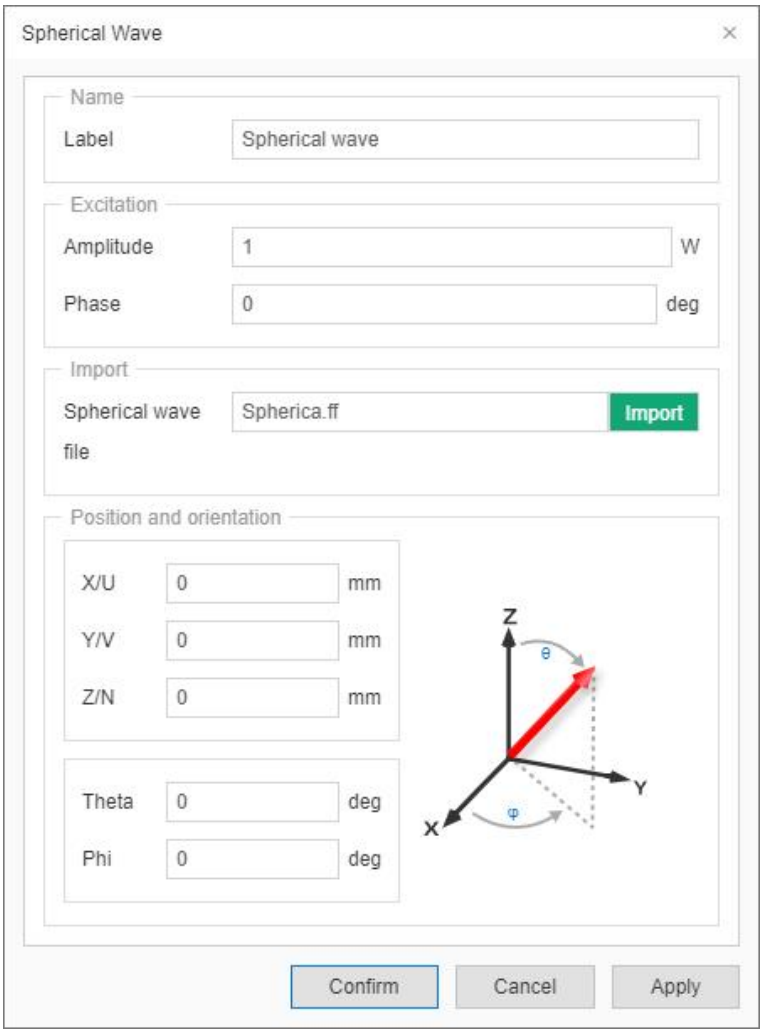


Figure 158 Spherical wave

5、 Click **Confirm** to finish.

6、 A spherical wave will appear in the viewing area.

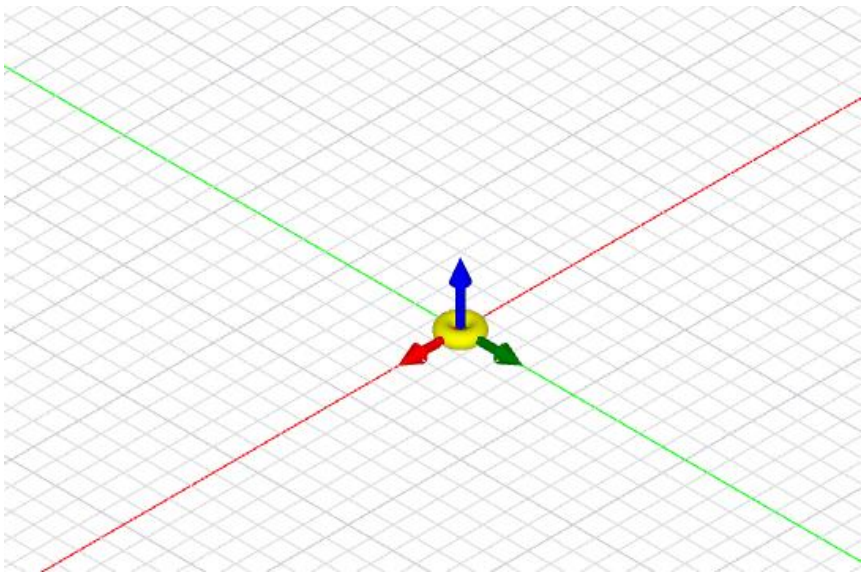


Figure 159 Spherical wave

7、 In the sub-node of the excitation in the project manager area, the spherical wave tag name will appear. Click the tag name, and a spherical wave setting window will pop up to be modified.

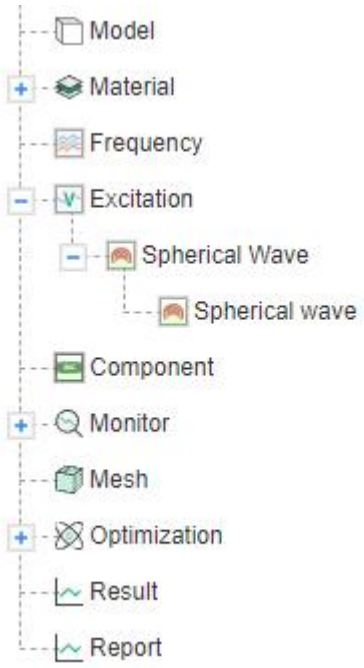


Figure 160 Spherical wave label name

5.5 Synthesis

Synthesis can realize the excitation synthesis between excitation sources.

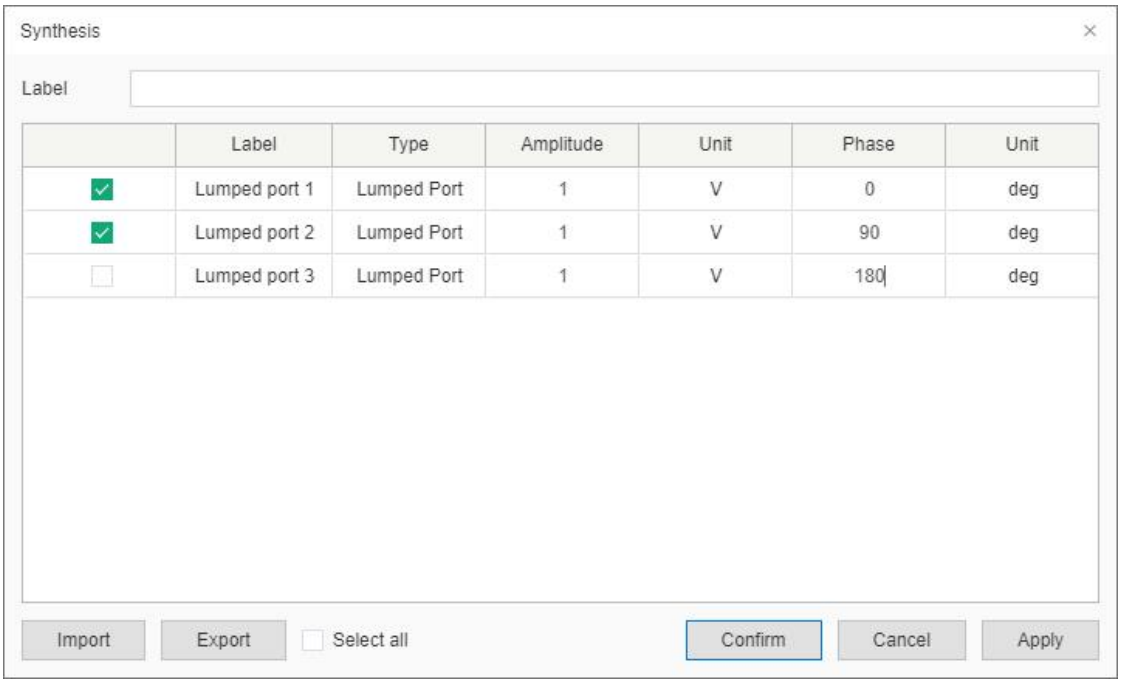


Figure 161 Incentive synthesis

The operation steps of incentive synthesis are as follows:

1、 Click the **Synthesis**.

2、 In the excitation synthesis window, enter the label name of the synthesis source after the name box.

3、 Check the selection box in the table to set the stimulus for synthesis, and input the amplitude and phase of each stimulus in turn under the amplitude and phase (the amplitude and phase can be imported by clicking, and the amplitude and phase can be imported from the file value).

4、 Click **Confirm** to finish.

5、 In the sub-node of Incentive in the project manager area, there will be an Incentive Synthesis-Incentive Synthesis tag name. Click on this tag name, a synthesis source window will pop up to be modified.

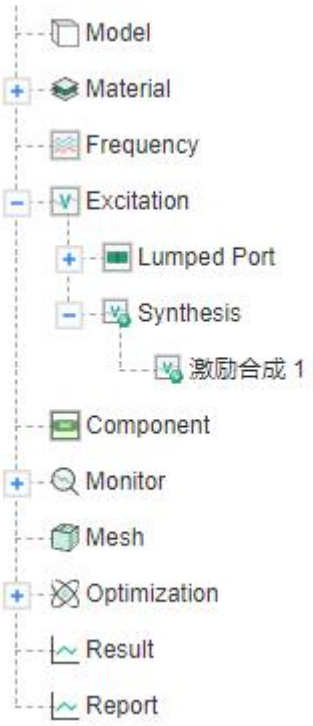


Figure 162 Incentivize synthetic label names

Note: Plane waves cannot be combined with other sources.

5.6 Components

5.6.1 Line Components

Line components assign passive components on a line model. In the line component setting, input the name of the target line and component name, select the adding position of the component, and choose the component type.

The operation steps of line component setting are as follows:

1、 Click **Line component**.

Figure 163 Select line components

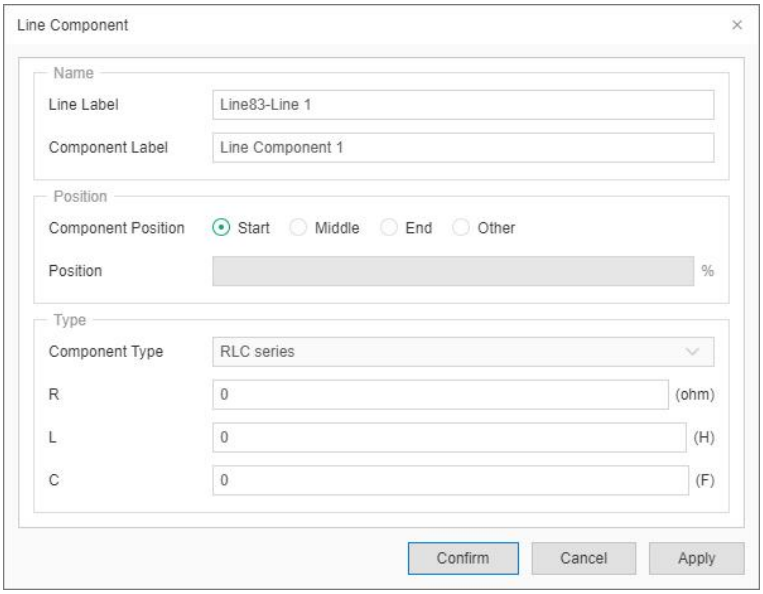


Figure 164 Line component

2、 In the line component window, first select the line model in the view area and or enter the line name.

3、 Enter the component name (the default is Line component 1, 2...).

4、 Choose component position under the position box. If other is chosen, enter the component position in the input box after the position (0-100, where 0 represents the starting point, 100 represents the end point).

5、 In the Type column, select RLC Series, RLC Parallel or Custom Impedance.

If RLC in series/parallel is chosen, input the values of R (resistance), L (inductance), and C (capacitance) to set the impedance when they are in series.

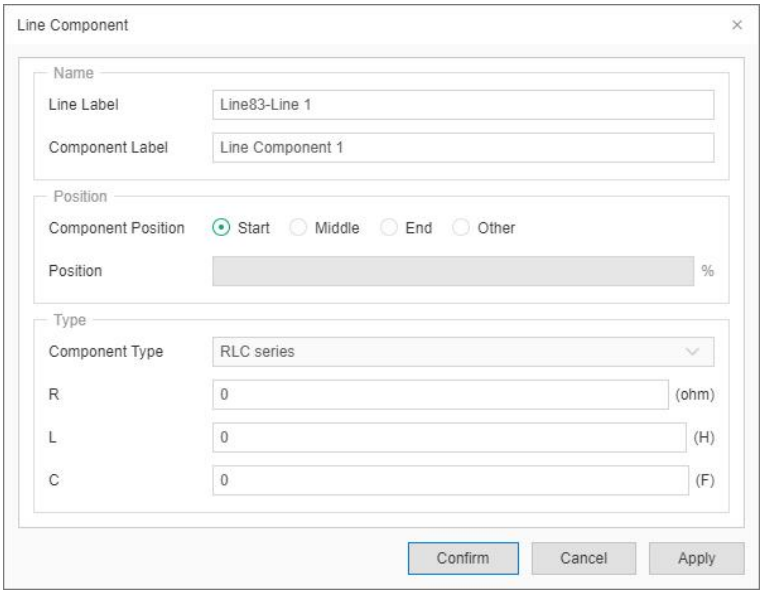


Figure 165 RLC series

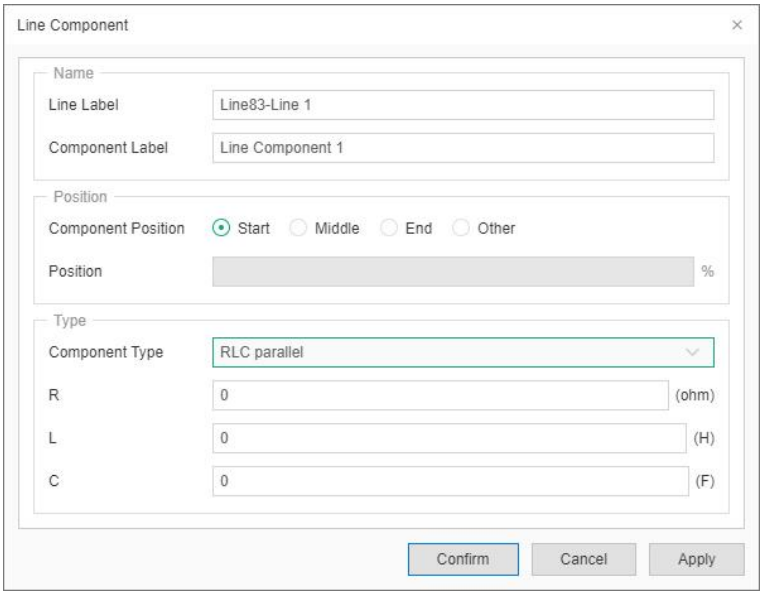


Figure 166 RLC parallel

If custom impedance is chosen, define the real part and imaginary part of the line component impedance, or import a load file to set the line component impedance.

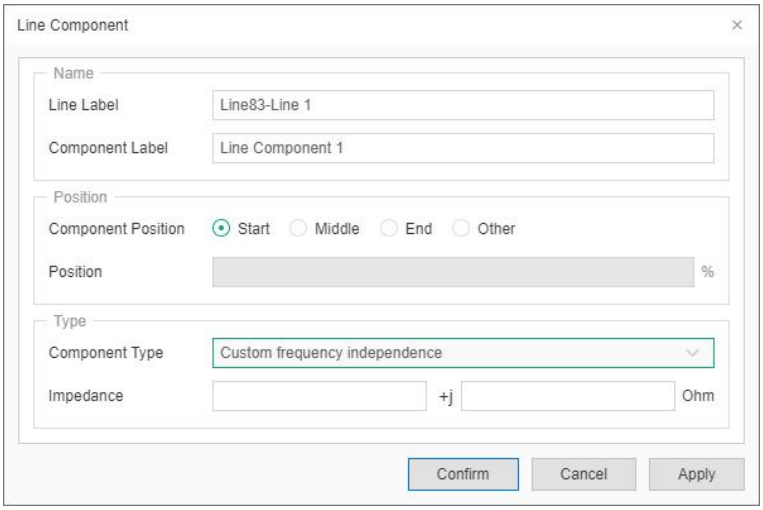


Figure 167 Customize

6、 Click **Confirm** to finish.

7、 A line component will appear on the line model in the view area.

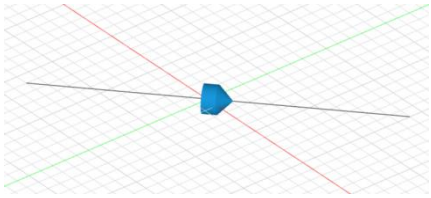


Figure 168 Line component

8、 In the sub-node of the project manager area, there will be a component - line component label name, click on the label name, and a property window for line component settings will pop up to be modified.

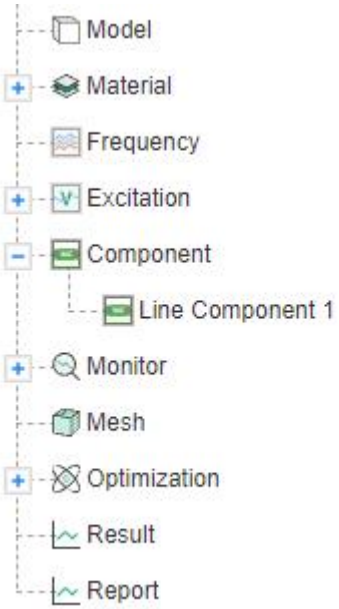


Figure 169 Line component label name

5.6.2 Boundary Components

Boundary Components assign passive components at the boundary between two surfaces. In the boundary component setting, input the component name, select the position of the component, and set the component type.

The operation steps of boundary element setting are as follows:

1、 Click **Boundary Component**.

Figure 170 Select boundary components

Figure 171 Boundary component

2、 In the boundary component window, select the line model on the boundary in the view area, and enter the line name.

3、 Enter the port name after the port name (the default value is Boundary component 1, 2...).

4、 Select component position under the position box, if you choose other, you need to enter the component position in the input box after the position (0-100, where 0 represents the starting point, 100 represents the end point, and the number in 0-100 represents the point position between the start and end of the line).

5、 In the Type column, select RLC Series, RLC Parallel or Custom Impedance.

If choose RLC in series is chosen, input the values of R, L, and C to set the impedance when RLC are in series.

Figure 172 RLC series

If RLC in parallel is chosen, input the values of R, L, and C to set up RLC parallel connection.

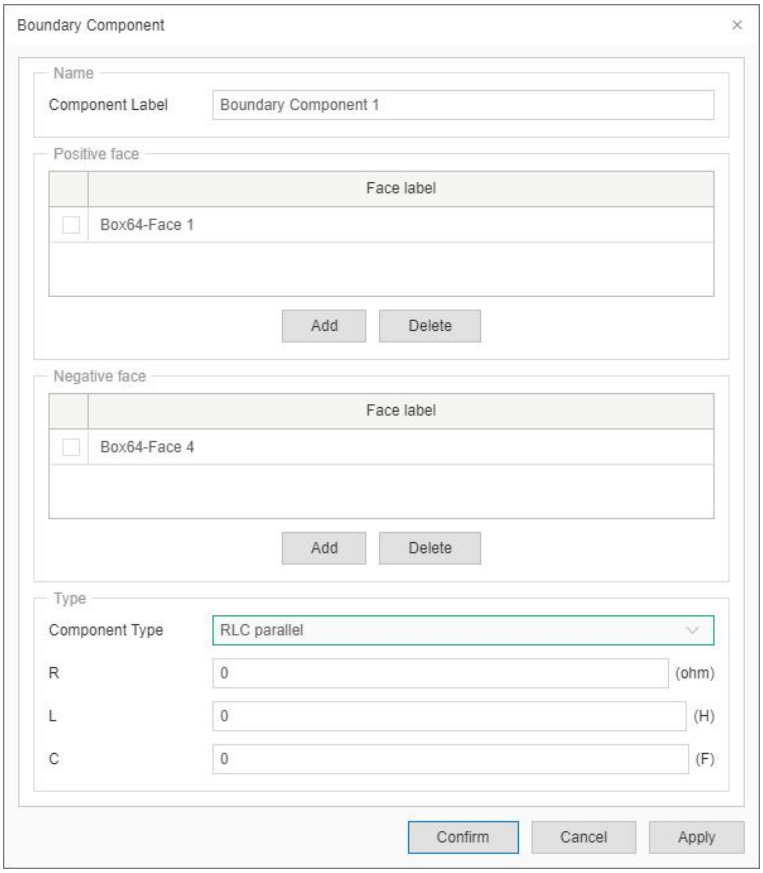


Figure 173 RLC parallel

If custom impedance is chosen, define the real part and imaginary part of the impedance or import the load file to define the impedance of the boundary components.

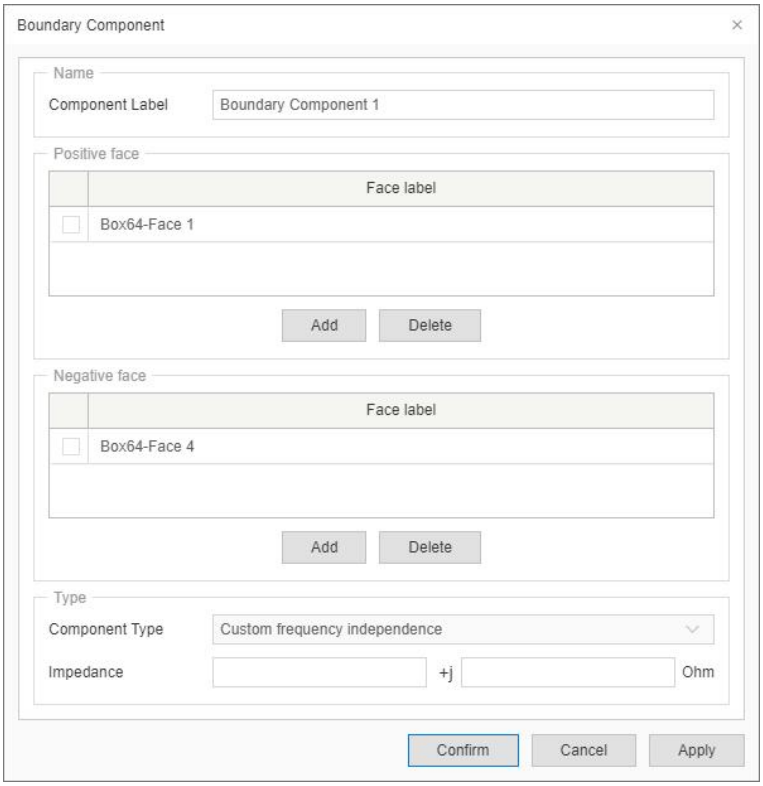


Figure 174 Customize

6、 Click **Confirm** to finish.

7、 A boundary component will appear on the selected boundary of the view area.

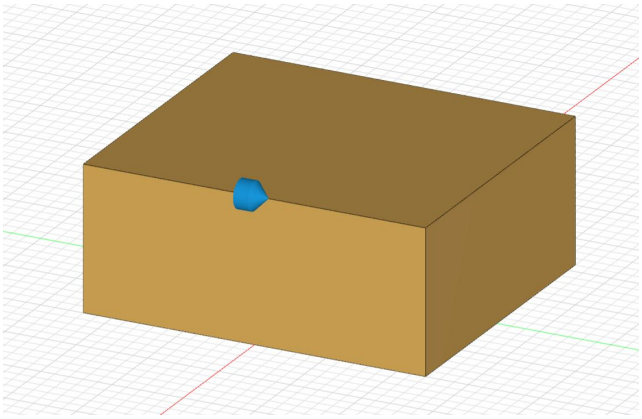


Figure 175 Boundary components

8、 In the sub-node of the project manager area, there will be a label name of component - boundary component. Click on the label name, a property window of will pop up to be modified.

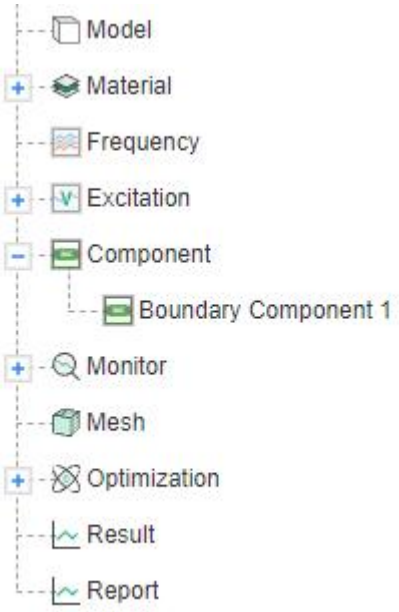


Figure 176 Boundary component label name

5.7 Monitor

Monitor includes **Current** result setting, **Near Field** result setting and **Far Field** result setting.

5.7.1 Current result settings

The current result settings set up the current monitor, where it can be chosen to calculate current of the whole model or part of the structure.

The operation steps of current result settings are as follows:

1、 Click **Current.**

2、 In the current result setting window, enter the label name of the current monitor (default as current+number).

3、 Choose solving surface in the current box : all or part

i. If select all, click **Confirm**.

Figure 177 Current results settings - all

4、 In the project manager area Result Setting-Current sub-node, the current result setting label name will appear. Click on the label name, and the current result setting window will pop up to be modified.

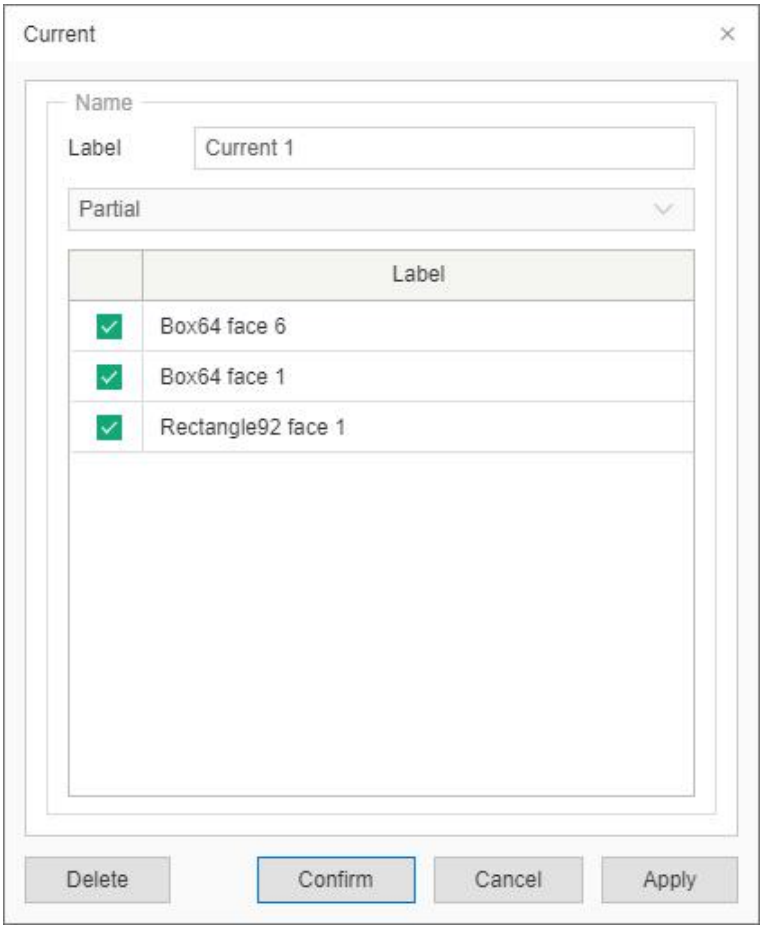


Figure 178 Current results setup -partial

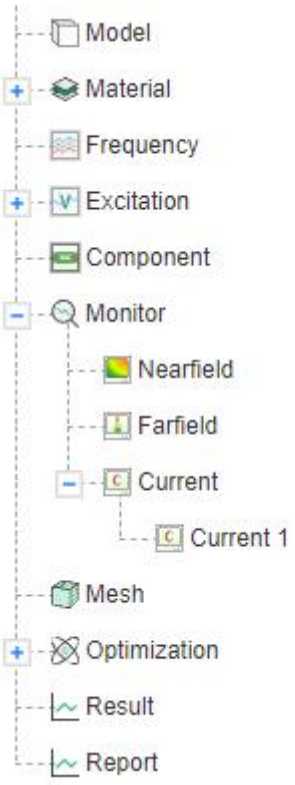


Figure 179 Current results set tab name

Note: The current can only be calculated on the metal surface.

5.7.2 Near Field Results Settings

The Near Field Results settings set up the near field monitor, where the user needs to enter a label name, select to calculate the electric field or magnetic field, enter the starting point, end point, and interval in the coordinate system table, select to calculate the total field or the scattered field, and select the solving surface (all or part).

The operation steps of near-field result setting are as follows:

1、 Click **Near-Field**.

2、 In the near-field result setting window, first enter the label name of the near-field result setting after the name box-label name (the default is near field + number).

3、 In the selection box on the left, uncheck the same direction of incidence.

4、 In the coordinate system box, enter the start point, end point, and interval of theta and phi in sequence.

5、 In the selection box on the right, select the total field.

6、 In the solve surface box select: all or part

i. If select all, click **Confirm** to complete the near-field result setting.

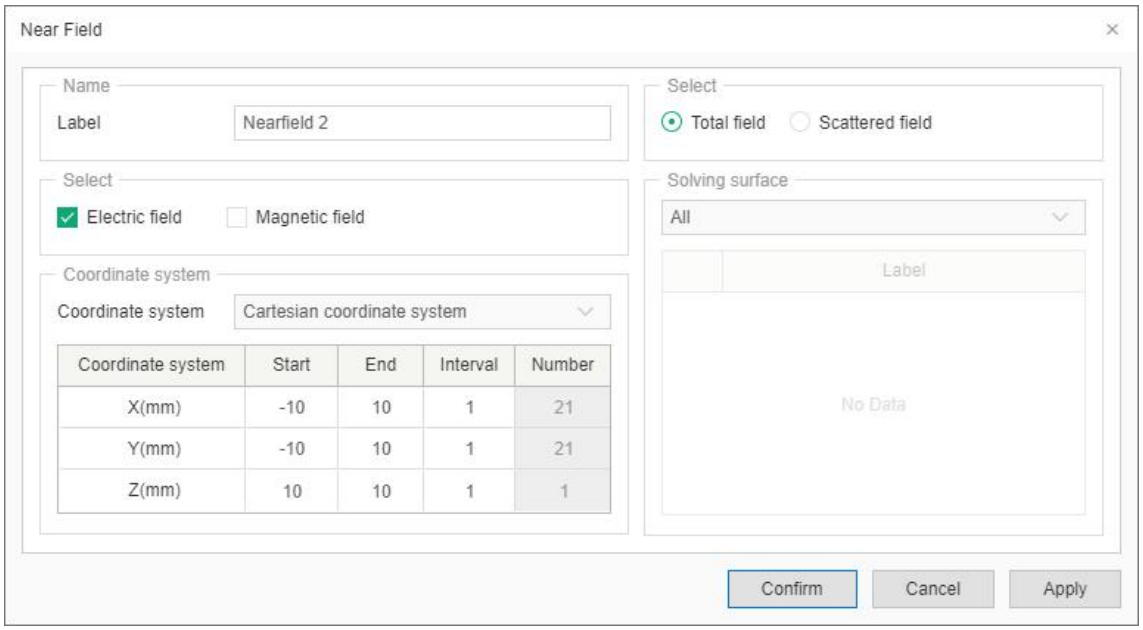


Figure 180 Near field results settings - all

ii. If you select a part, move the mouse to the view area and click the solution surface. To delete, move the mouse to click the label name of the surface, and then click delete. Finally click **Confirm** to complete finish.

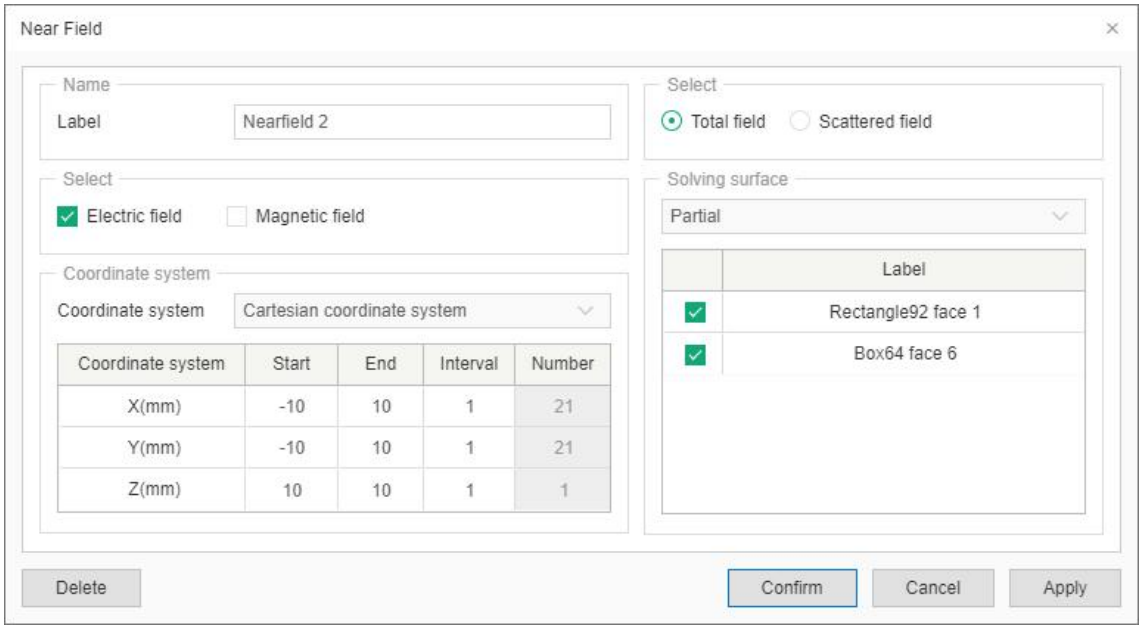


Figure 181 Near field results settings - partial

7、 In the view area, the near-field result settings are displayed.

8、 The near-field result setting tab will appear under the result setting-near-field sub-node in the project manager area. Click the tab name, and a near-field result setting window will pop up to be modified.

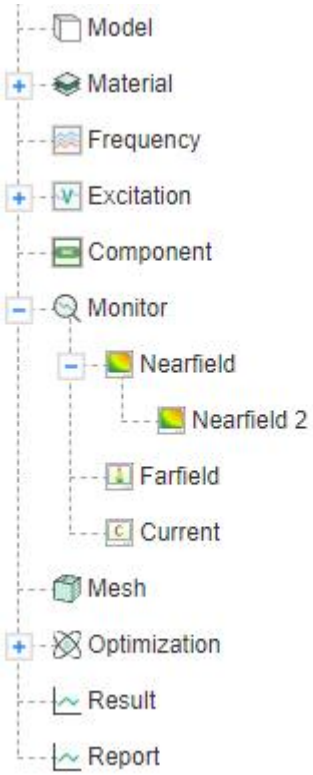


Figure 182 Near field results settings label name

5.7.3 Far Field Result Settings

The Far Field Result settings set up the far field monitor. In the far-field result settings, the user needs to input the label name, select whether it is in the same direction as the incident wave, (far field monitor in the same direction as the incident wave calculates single-station RCS), enter the start point, end point and interval in sequence in the coordinate system table, choose to calculate the total field or scattered field, and choose the solving surface (all or part).

There are two application scenarios for the far-field result setting, one is used to calculate far-field results, and the other is used to calculate single-station RCS. The following describes the setting methods and steps of the far-field results in the two application scenarios.

5.7.3.1 Far Field Results Settings

The operation steps of far-field result setting are as follows:

1、 Click **Far Field**.

2、 In the far field result setting window, enter the label name of the far field result setting in the name box (default as far field+number).

3、 In the Single Station RCS box, uncheck Same Incidence Direction.

4、 In the coordinate system box, input the start point, end point, and interval of theta and phi in sequence.

5、 field component selection box, select the total field.

6、 In the solve surface box select: all or part.

i. If select all, click **Confirm** to finish.

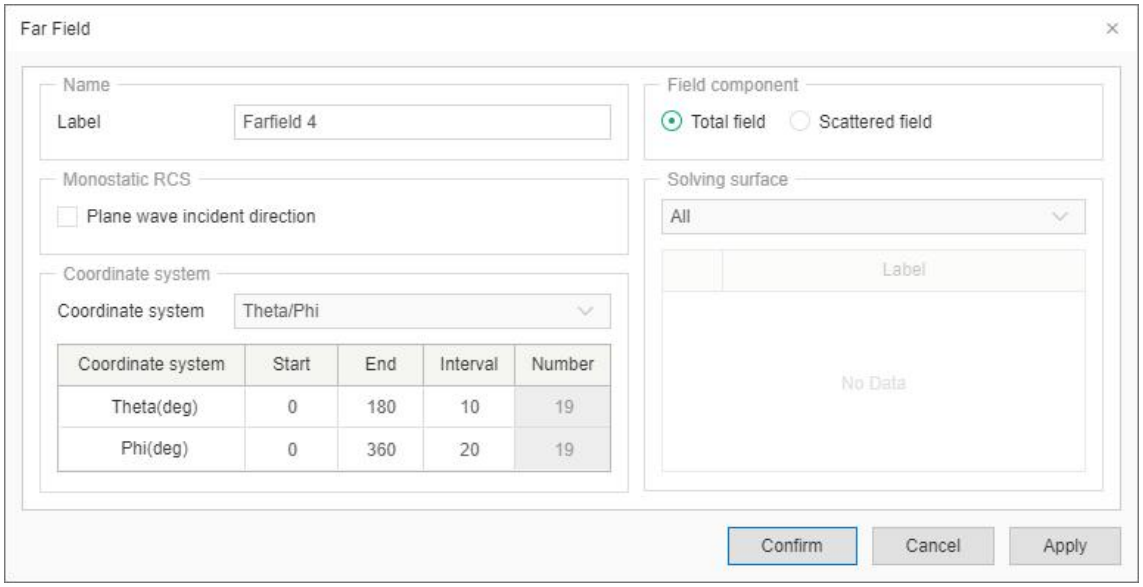


Figure 183 Far field results settings - all

ii. If select a part, move the mouse to the view area, and click the solving surface. To delete, move the mouse to click the label name of the surface, and then click **Delete**. Finally click **Confirm** to finish.

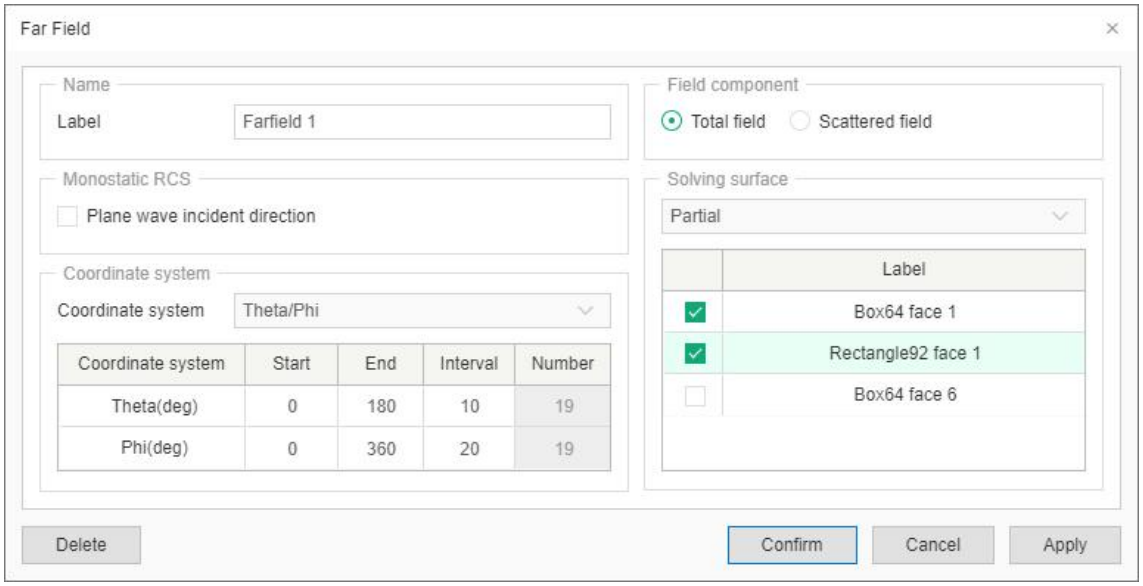


Figure 184 Far field results settings - partial

7、 In the view area, the far-field result settings are displayed.

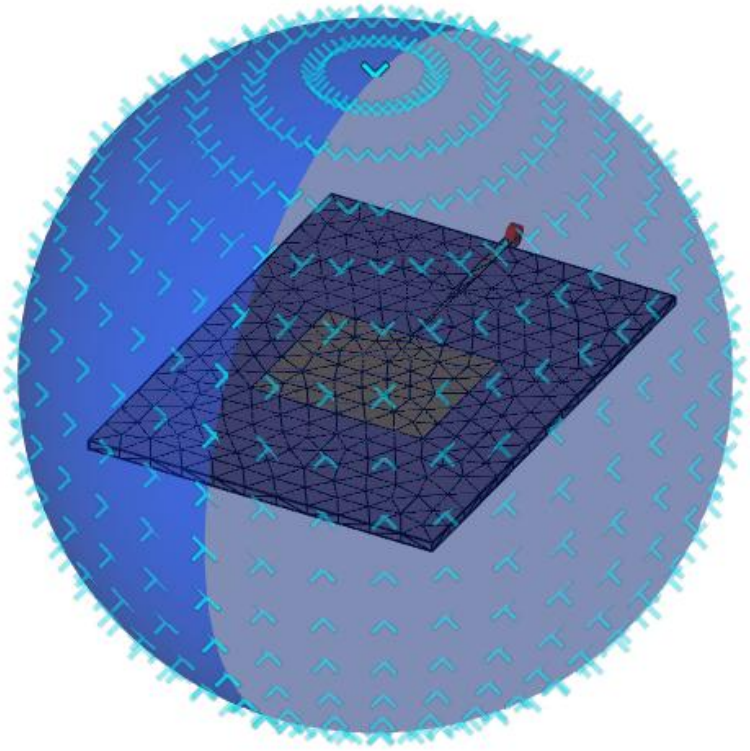


Figure 185 Far field results settings

8、 The far-field result setting tab will appear under the Result Setting-Far Field sub-node in the project manager area. Click on the tab name, and a far-field result setting window will pop up to be modified.

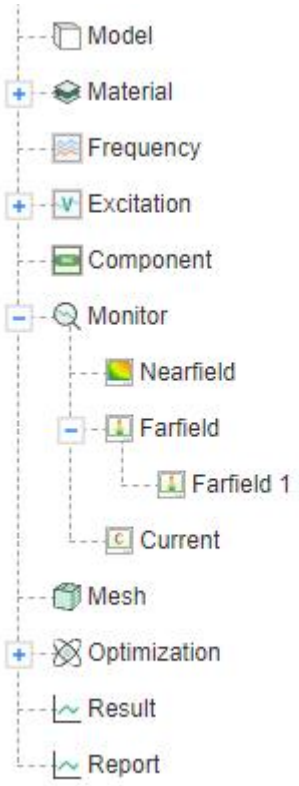


Figure 186 Far field results settings tag name

5.7.3.2 Single station RCS far field result setting

Before setting up a single-station RCS far-field monitor, there must be a plane wave excitation in the project, otherwise an error will be reported.

The operation steps of single-station RCS far-field result setting are as follows:

1、 Click **Far-Field**.

2、 In the far field result setting window, enter the label name of the far field result setting in the name box (the default is far field momiter + number).

3、 In the single-station RCS box, check the same direction of incidence.

4、 Field Components box, select Total Field.

5、 In the solve surface box select: all or part

i. If select all, click **Confirm** to finish.

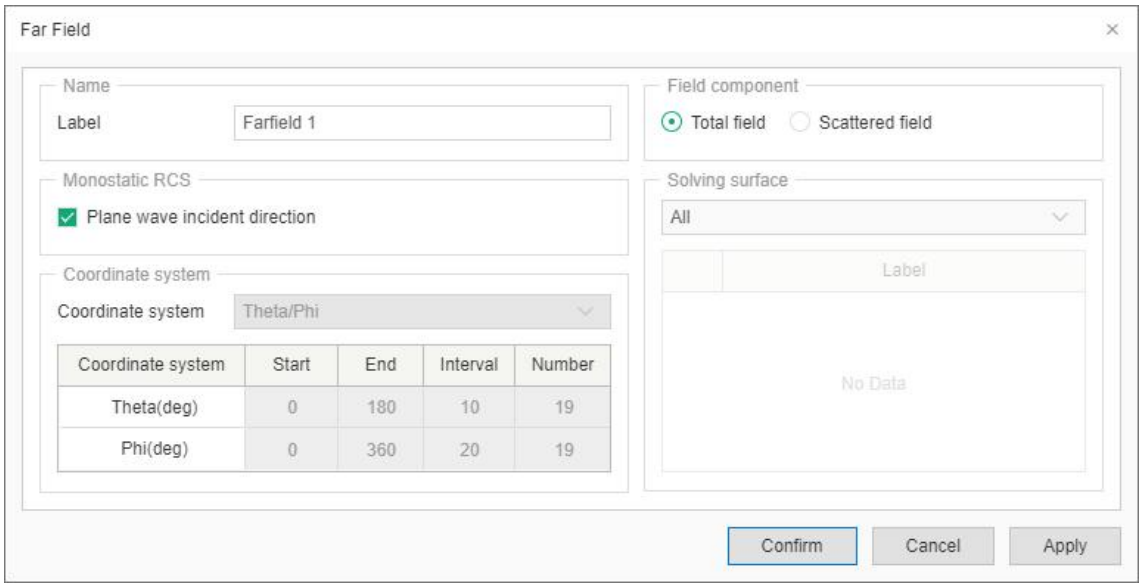


Figure 187 Single station RCS far field result settings - all

ii. If select a part, move the mouse to the view area, and click the solution surface. To delete, move the mouse to click the label name of the surface, and then click **Delete**. Finally, click **Confirm** to finish.

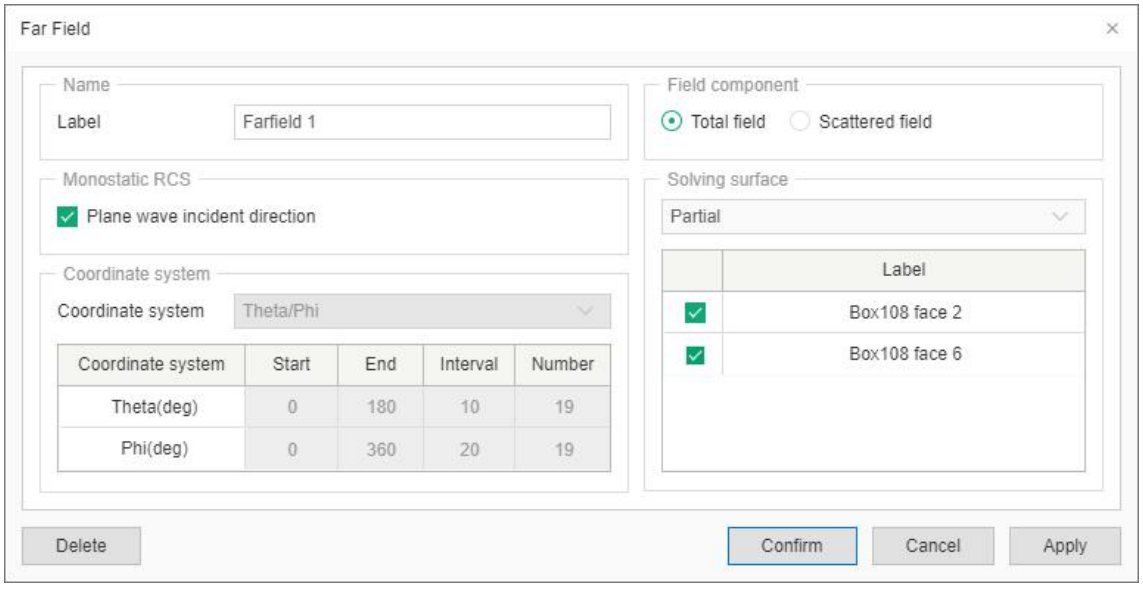


Figure 188 Single station RCS far field result settings - partial

6、 In the view area, the far-field result settings are displayed.

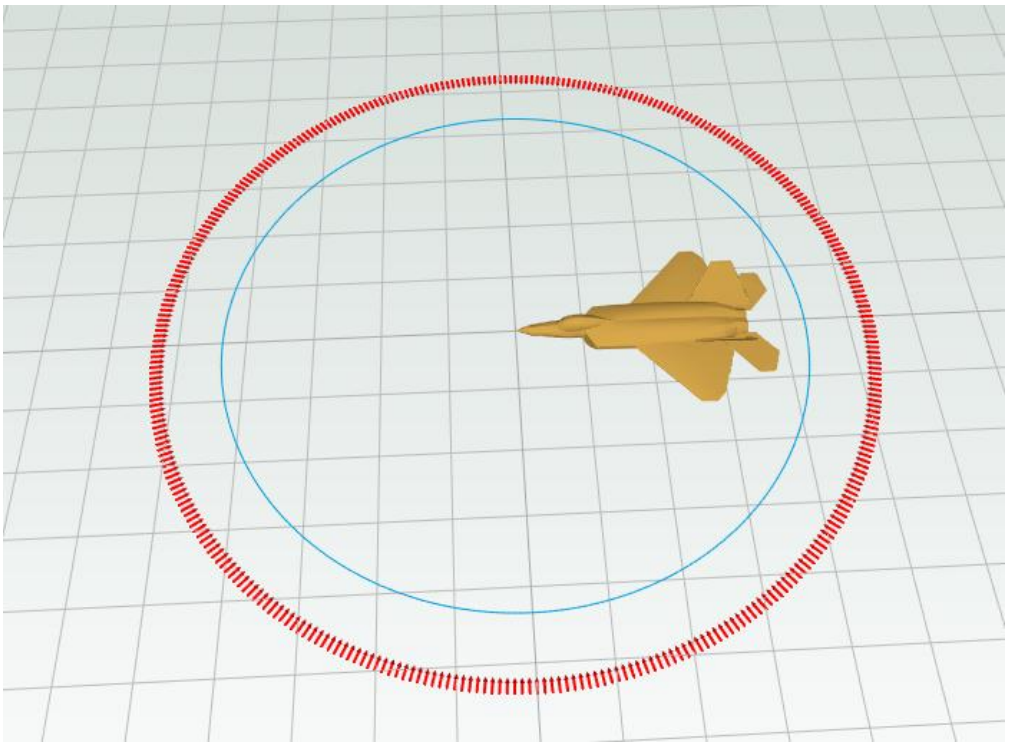


Figure 189 Single station RCS far field result setting

7、 In the project manager area Result Setting-Far Field sub-node, the Far Field Result Setting tab will appear. Click on the tab name, and a far-field result setting window will pop up to be modified.

6 Meshing Module

Mesh Module includes **Create Mesh, Mesh Check, Refinement, Mesh Information** and **Mesh Cleanup**.

6.1 Create mesh

Click **Create Mesh** to mesh the model. In the mesh window, select mesh size (sparse, standard, coarse). If the project includes line models, wire segment radius of the line model needs to be entered.

The mesh operation steps are:

1、 Click **Create Mesh**.

2、 In the mesh window, select all.

3、 Select the mesh size in the global mesh size: sparse, standard, coarse or custom.

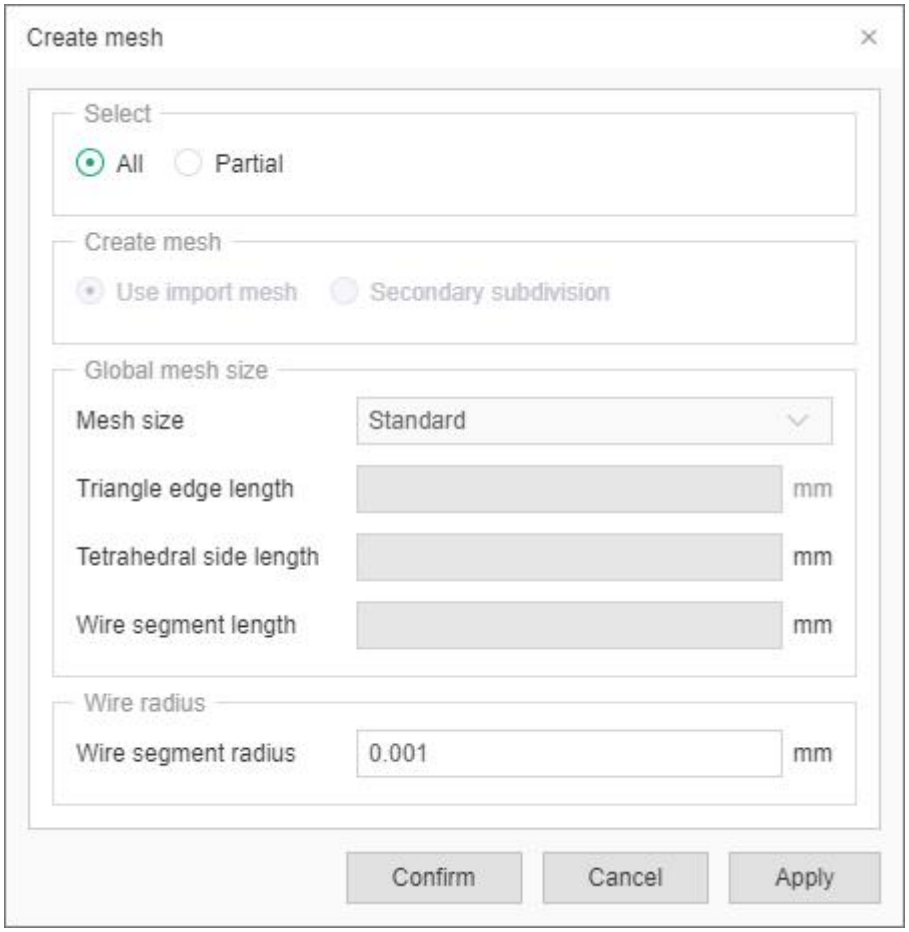


Figure 190 Mesh

4、 If a custom mesh size is selected, enter the custom mesh size of the triangulation edge length, tetrahedron side length and thin line length.

5、 Enter the radius of the thin line in the mesh size box of the online model.

6、 Finally, click **Confirm** to mesh.

7、 After the meshing is completed, the mesh view will be displayed in the view area.

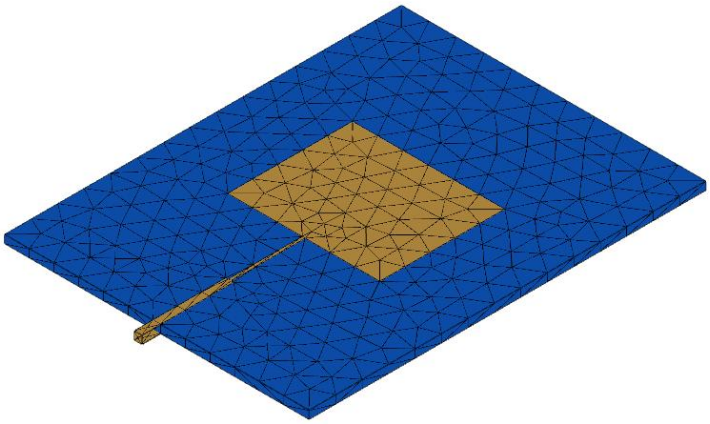


Figure 191 Mesh view

8、 In the project manager area - mesh, the mesh label name will appear, click the right mouse button to perform delete, show/hide operations.

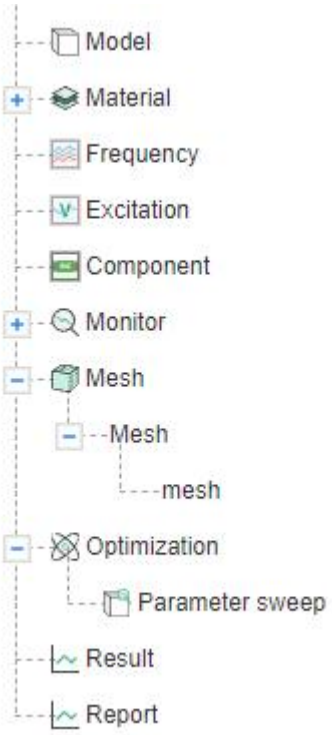


Figure 192 Mesh tag name

6.2 Mesh Check

Mesh check includes **Intersecting Models**, **Intersecting Triangles**, **Distorted Elements** and **Oversized Elements**.

6.2.1 Intersecting Models

Intersecting model detection detects whether there are intersections between models. If there are, the intersecting model tag numbers will be displayed.

The operation steps of intersecting model detection are as follows:

1、 Click **Mesh Check**.

2、 Select **Intersecting Models** in the drop-down box, and the intersecting model detection setting window will pop up.

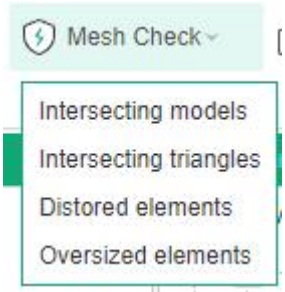


Figure 193 Mesh check

3、 In the pop-up window, click **Confirm** to perform detection.

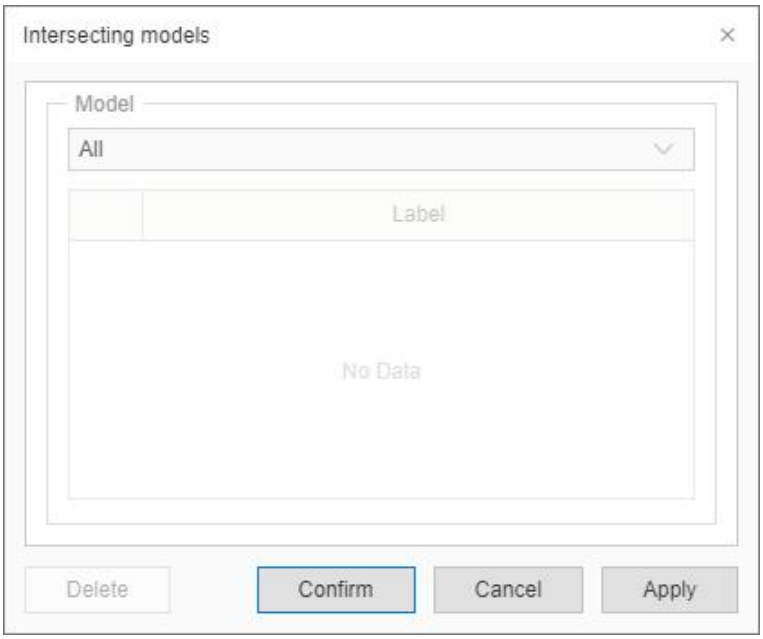


Figure 194 Intersecting model

4、 After the detection, the results will be displayed in the information manager area.

6.2.2 Intersecting Triangles

Intersecting triangles detection detects whether there are intersections between triangular meshes. If there are, the tag numbers of the intersecting triangles and the surface where it is located will be printed in the information manager area.

The operation steps of intersecting triangle mesh detection are as follows:

1、 Click **Mesh Check**.

2、 Select intersecting triangular in the drop-down box, and a window for Intersecting Triangular Detection Settings will pop up.

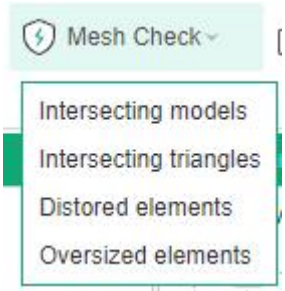


Figure 195 Mesh check

3、 In the detection setting window, click **Confirm** to perform detection.

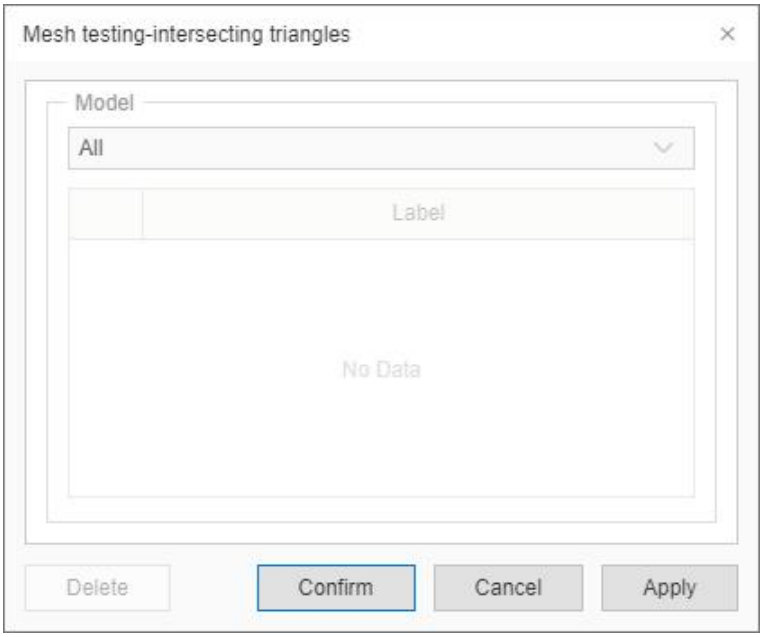


Figure 196 Intersecting triangle

4、 After the detection, the results will be displayed in the information manager area.

6.2.3 Distorted Elements

Distorted elements detection detects whether there are distorted units in the triangular meshes (one of the interior angles of some triangles are particularly large). If there are, the tag numbers of the distorted unit and the surface where it is located will be printed in the information manager area.

The operation steps of distorted unit detection are as follows:

1、 Click **Mesh Check**.

2、 Select the distorted elements in the drop-down box, and the detection setting window will pop up.

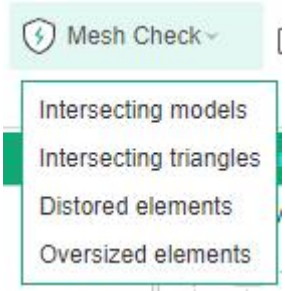


Figure 197 Mesh check

3、 In the detection setting window, enter the minimum inner angle in the input box, and click **Confirm** to perform detection.

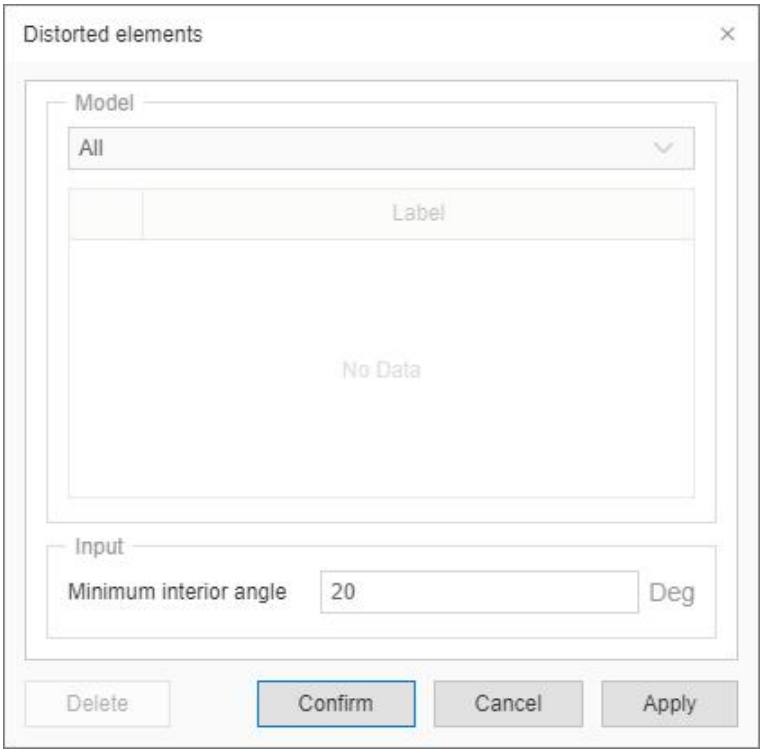


Figure 198 Deformation elements

4、 After the detection, the result will be displayed in the information manager area.

6.2.4 Oversized Elements

The oversized elements detection detects whether there are triangle mesh lengths are too long. If there are, the tag numbers of these elements and the surfaces where it is located will be

printed in the information manager area.

The operation steps of intersecting triangle mesh detection are as follows:

1、 Click **Mesh Check**.

2、 Select the oversized elements in the drop-down box, and detection setting window will pop up.

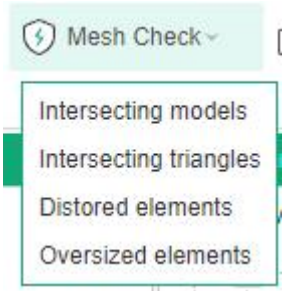


Figure 199 Mesh check

3、 In the window, enter length in the input box, click **Confirm**, and perform overlong unit detection.

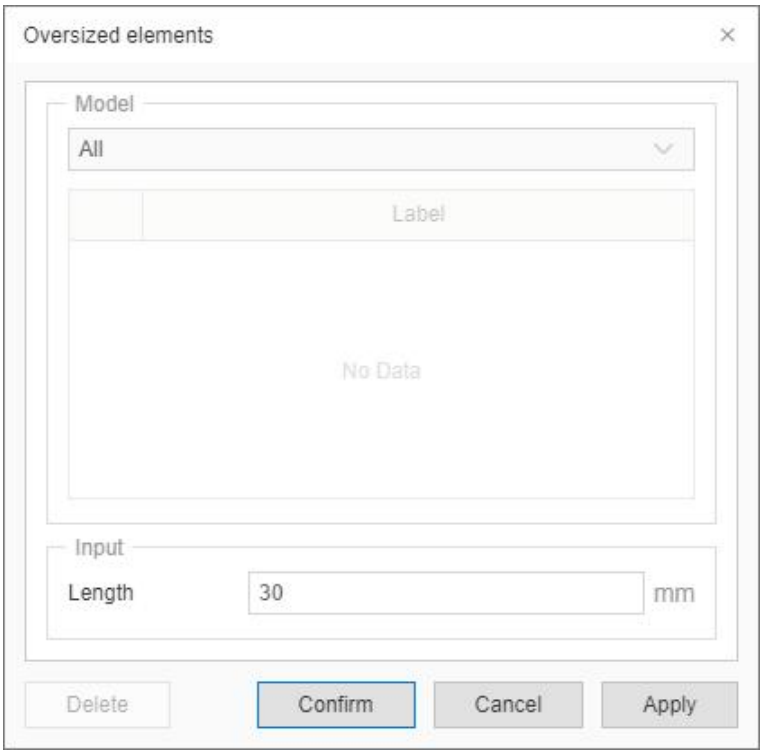


Figure 200 Oversized elements

4、 After the detection, the result will be displayed in the information manager area.

Note: Intersecting triangular mesh detection, deformation element detection, and overlong element detection must be performed after meshing.

6.3 Refinement

Refinement locally remeshes the model. Local refinement includes point refinement, line refinement and local mesh size, etc.

6.3.1 Point Refinement

Point refinement locally refines mesh around a certain point. In point refinement, the user needs to input the coordinate position, refinement radius and refinement size of the refinement point.

The operation steps of point refinement are as follows:

1、 Click **Refinement.**

2、 Select point refinement in the drop-down box, and a point refinement setting window will pop up.

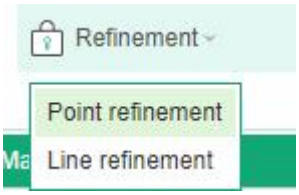


Figure 201 Local refinement

3、 In the pop-up window, enter the label name (default point refinement+number) after the name box - label name,

4、 Enter the x, y, z coordinate information of the encrypted point in the input (or move the mouse, click the left button of the mouse in the view area to obtain the coordinate information), radius and mesh size.

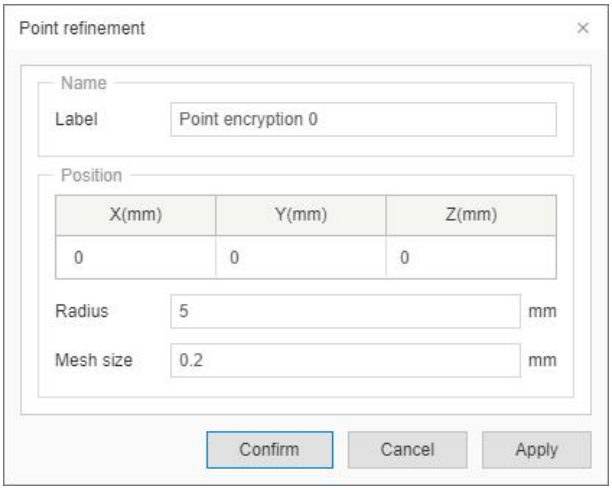


Figure 202 Point refinement

5、 The point refinement label name will appear under the division of the project manager area - point refinement, click on the label name, and a point refinement label setting window will pop up to be modified.

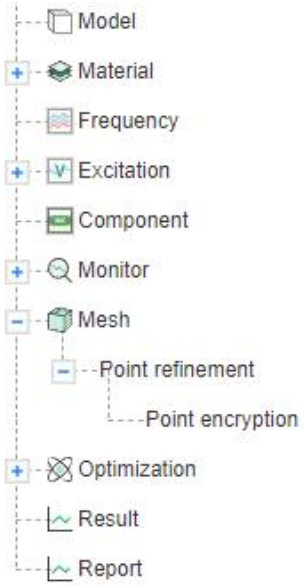


Figure 203 Point refinement label name

6.3.2 Line Refinement

Line refinement locally refines around a certain line. In line refinement, input the vertex coordinate position, refinement radius and refinement size of the line.

The operation steps of line refinement are as follows:

1、 Click **Refinement**.

2、 Select line refinement in the drop-down box, and a line refinement setting window will pop up.

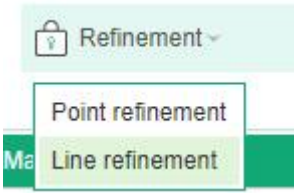


Figure 204 Local refinement

3、 In the pop-up window, enter the label name (default line refinement+number) after the name box - label name,

4、 Input the x, y, z coordinate information of the point on the refinement line in the input (or move the mouse and click the left button of the mouse in the view area to obtain the coordinate information), radius and mesh size.

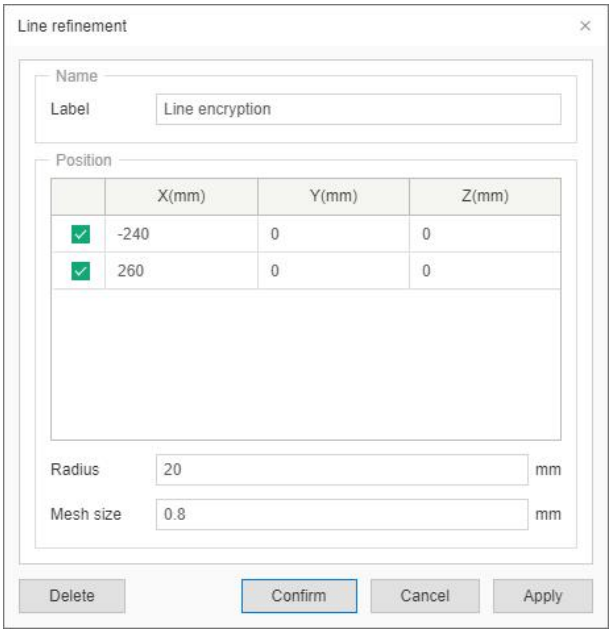


Figure 205 Line refinement

5、 The line refinement label name will appear under the division of the project manager area - point refinement, click on the label name, and a line refinement label setting window will pop up to be modified.

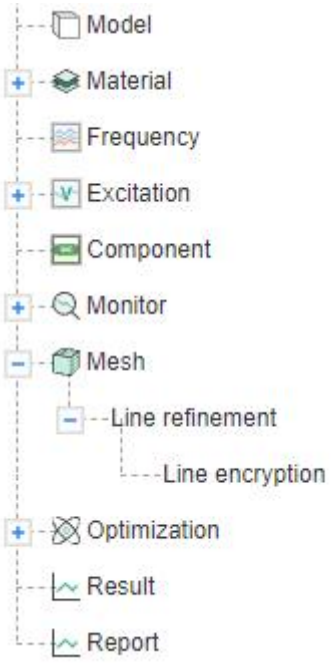


Figure 206 Line refinement label name

6.3.3 Local Mesh

Local mesh size setting sets the mesh size separately for a surface or body of the model.

a) The operation steps of surface mesh size setting are as follows:

1、 Select a model and select the name tag of one or multiple surfaces in property manager, click the right mouse button, and click property.

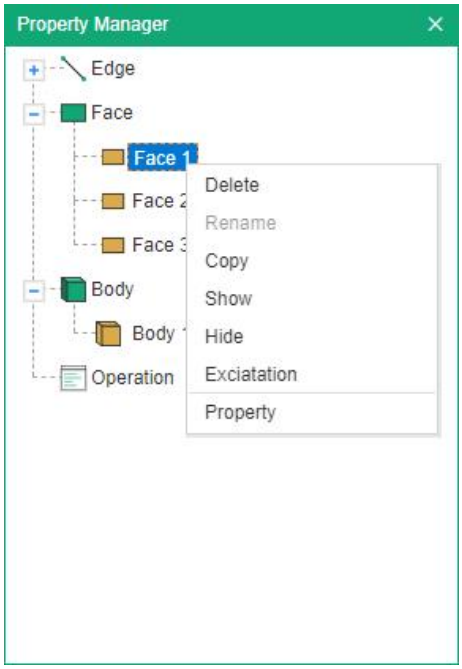


Figure 207 Property manager

2、 In the property window, check Use local mesh, and enter the mesh size.

3、 Click **Confirm** to complete the setting of a surface mesh size.

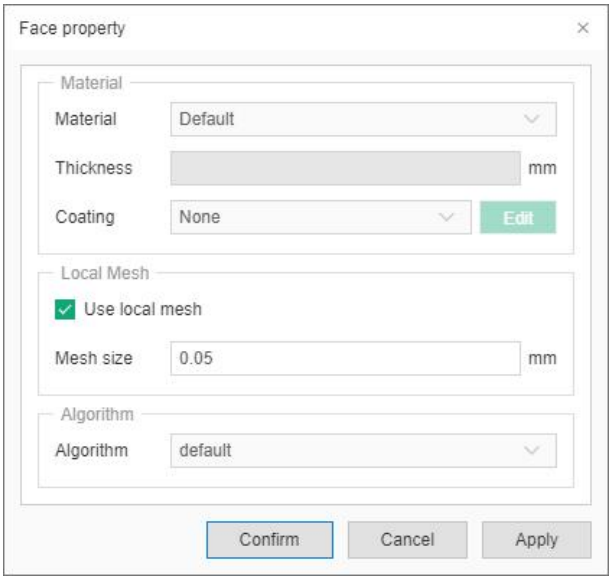


Figure 208 Surface mesh size

b) The operation steps of volume mesh size setting are as follows:

1、 Select a model and select the name tag of one or multiple bodies in property manager, click the right mouse button, and click property.

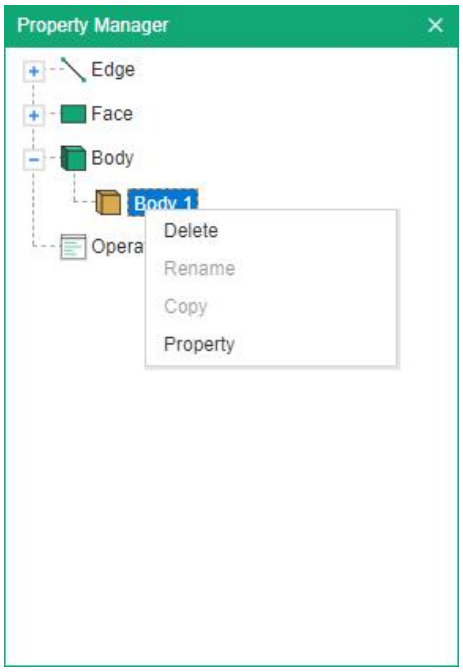


Figure 209 Property manager

2、 In the property window, check Use partial mesh size, and enter the mesh size value.

3、 Click **Confirm** to complete the volume subsection size setting.

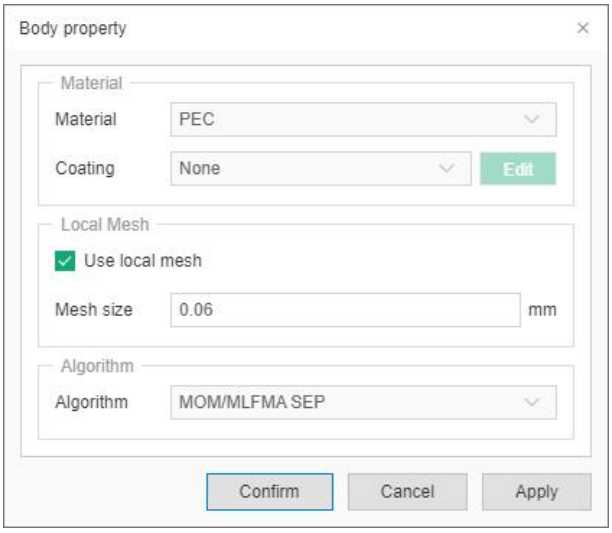


Figure 210 Body mesh size

6.4 Mesh Information

Click **Mesh information** to view the average size length, standard deviation of size length, mesh number, maximum size length and minimum size length of the mesh.

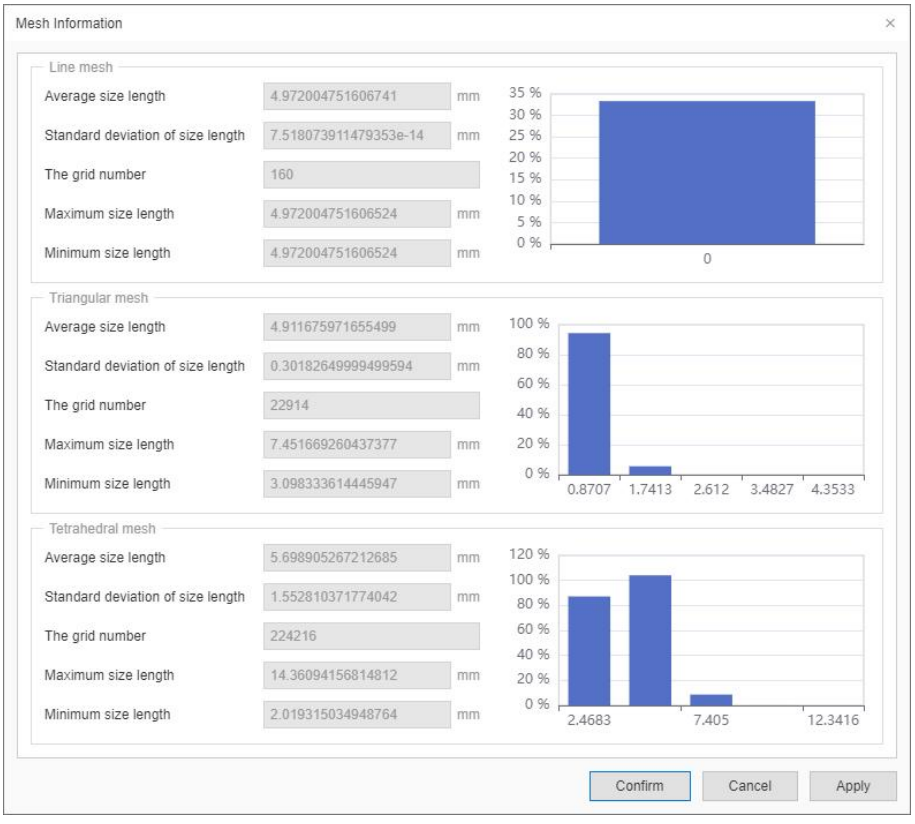


Figure 211 Mesh information

6.5 Mesh Cleanup

Click **Mesh cleanup** removes overlapping triangles in mesh.

7 Simulation module

The simulation module includes **Validate, Algorithm Settings** and **Run**. Options include **Hardware Options, HPC,** and **Remote**.

7.1 Validate

Validate function: Sweep parameters, models, incentives, segmentation, result settings, etc.

7.2 Parameter Sweep

Parameter sweep performs parametric scanning for the project.

Steps of scanning parameters are as follows:

1、 Click **Parameter Sweep**.

2、 Click **Add**.

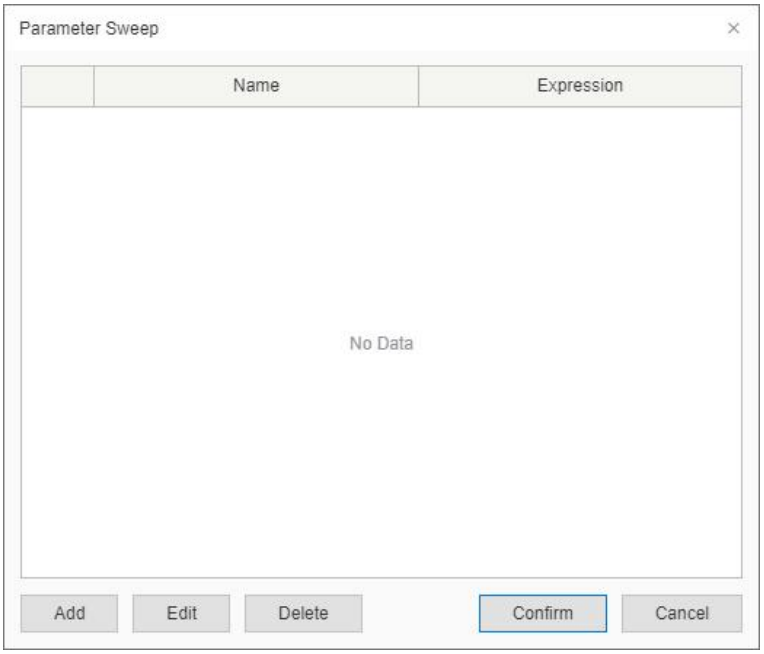


Figure 212 Parameter sweep

3、 In the Add Variable window, select variables that need to be scanned, select a interval method, and enter the starting point, end point and interval of the scanning parameter.

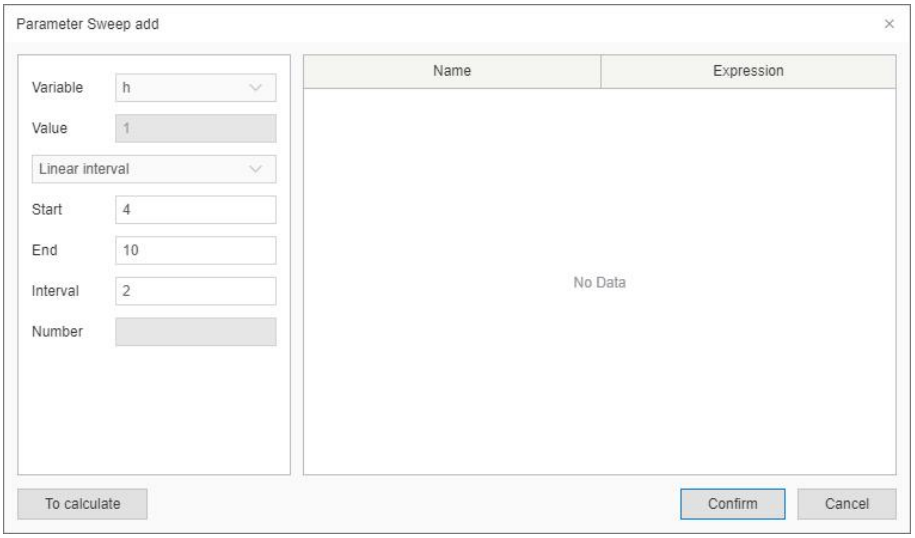


Figure 213 Add variable

4、 In the project manager area - Optimization - Scan Parameters, the name of the parameter scan tab will appear. Right click on the label name and click Run Scan Parameters to start the calculation of the parameters.

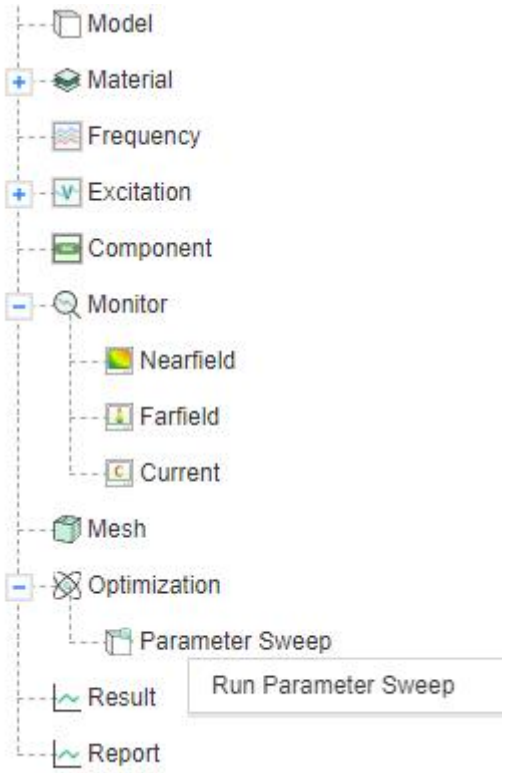


Figure 214 Run parameter sweep

7.3 Algorithm settings

Algorithm settings are specific settings for **MOM, MLFMA,** and **High Frequency** algorithms.

7.3.1 MOM Settings

In MOM algorithm settings, the solution precision includes single precision and double precision (default). Integral equations are disabled by default.

7.3.2 MLFMA algorithm settings

MLFMA algorithm settings include MLFMA accuracy, integral equation, iterative solver, preprocessing and bottom cube side length.

The solution precision includes single precision and double precision (default).

Stage precision defaults to 1.

Interpolation end defaults to 5.

Integral equations are disabled by default.

Iterative solvers include GMRES, TFQMR, and BICG.

If GMRES is selected, a number of restarts needs to be entered: default 300. residual: default 0.003. maximum number of iterations: 1500.

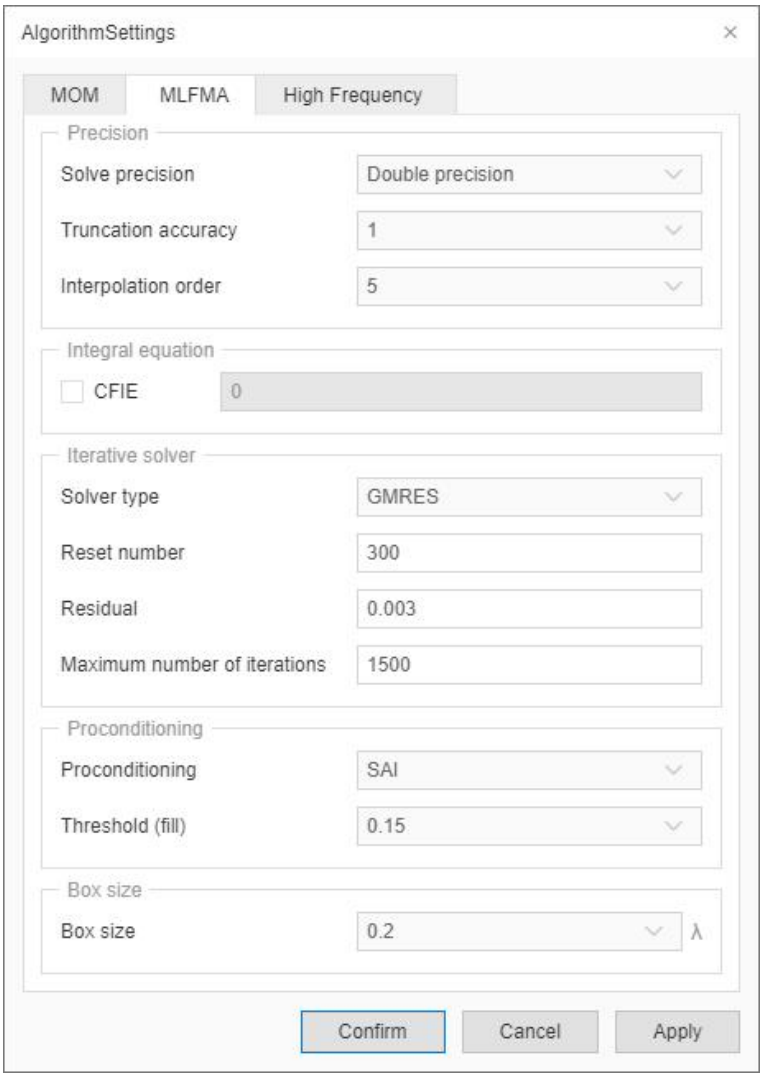


Figure 215 Algorithm settings - GMRES

If the solver type is selected as TFQMR: residual: default 0.003. maximum number of iterations: default 1500.

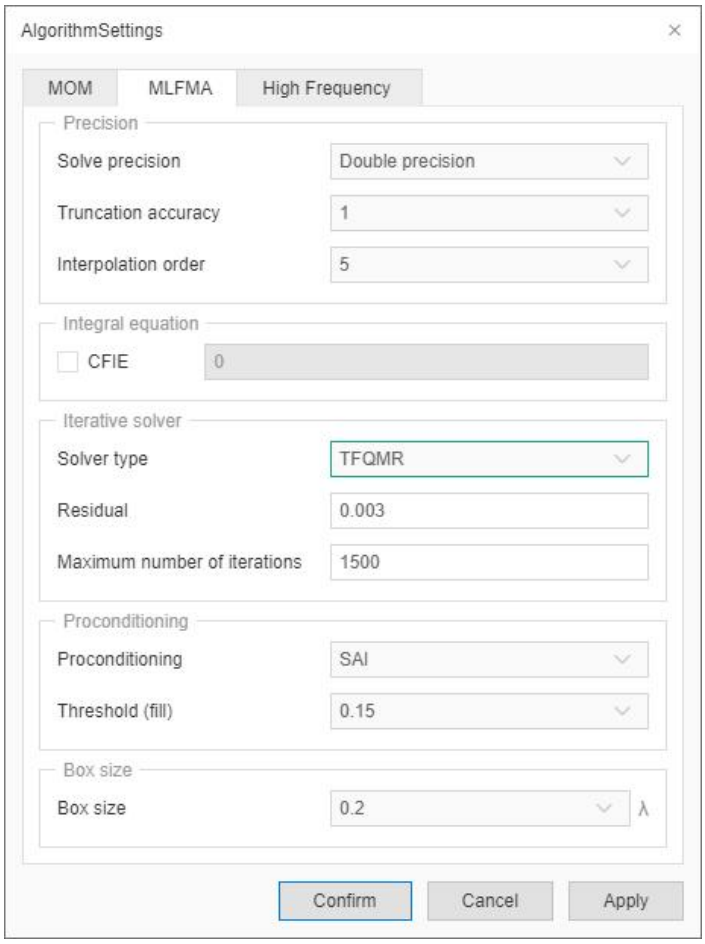


Figure 216 Algorithm settings - TFQMR

If the solver type is BICG: residual: default 0.003. maximum number of iterations: default 1500.

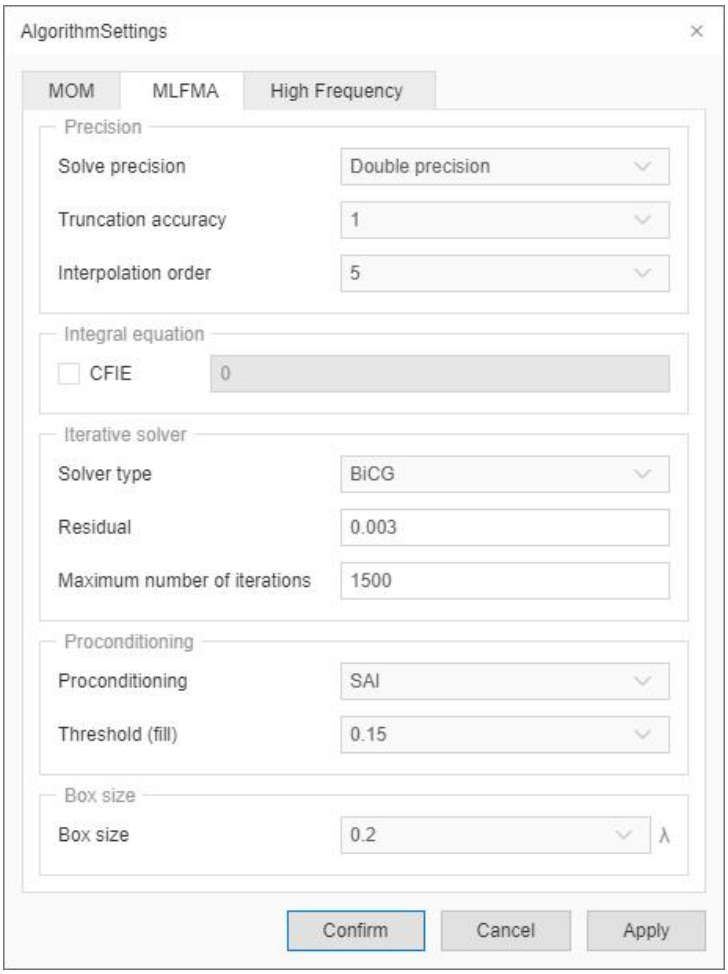


Figure 217 Algorithm settings - BiCG

The preconditions include SAI and ILU, and the default is SAI. the threshold is 0.15 by default.

The bottom cube side length defaults to 0.2, where λ is the vacuum wavelength.

7.3.3 SBR+ algorithm setting

SBR+ algorithm settings, including ray density, maximum number of bounces, and whether to calculate creeping waves.

Ray density settings include dense, standard, sparse and custom settings. In the case of custom selection, you need to enter a custom value in the box below. Enter single precision and double precision separately, the default is double precision.

The maximum number of bounces defaults to 1.

Calculate creeping waves is unchecked by default.

7.4 Run

Click the run button, and the run window will pop up. In the run window, users can set up the solution algorithm, excitation, and export options.

The operation steps to run are:

1、 Click the **Run**.

2、 In **Run** window, first select the solution method after the solution setting - solution method, the solution method includes MOM, MLFMA_SEP, MLFMA_VEP and High Frequency.

3、 Select the stimulus source after Excitation - Excitation Source:

i. All is selected by default.

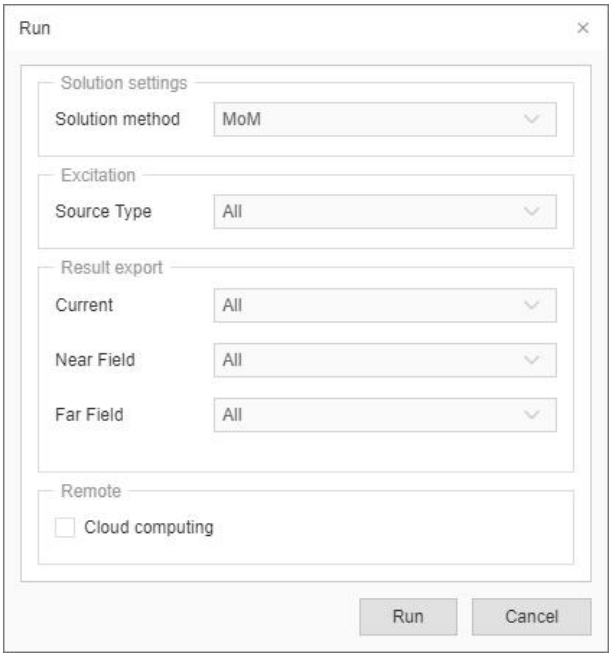


Figure 218 Run-all

ii. If you choose to customize, click the edit box, and in the source window, select the desired source tag names.

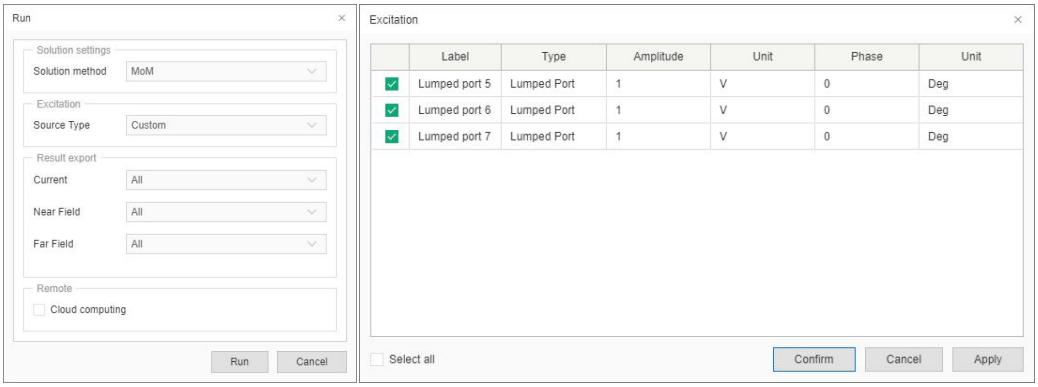


Figure 219 Run-excitation-custom

4、 In the result export box, you can choose to export the label names of current results, near field results and far field results (if not selected, none will be exported).

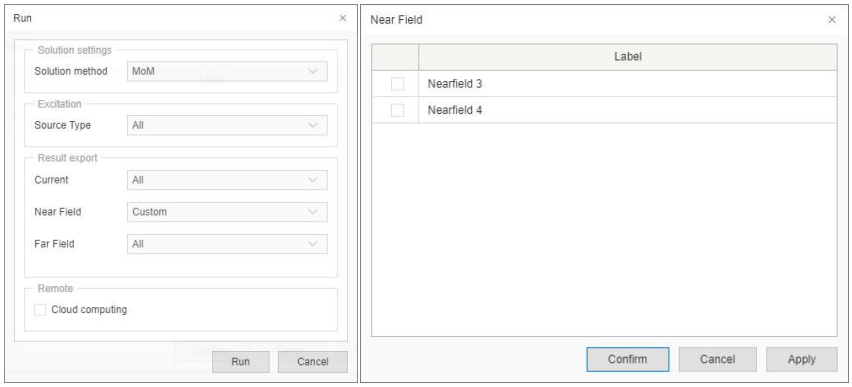


Figure 220 Run-near field-custom

5、 Click **Run** to calculate.

6、 If cloud computing is checked, the project will be submitted to the cloud server for simulation.

7.5 Options

7.5.1 Hardware options

In the hardware option, CPU or GPU can be selected for calculation, and the number of CPU cores and GPUs can be modified.

The hardware option operation process is as follows:

1、 Click the **Hardware Options** .

2、 In the hardware option window, check the number of CPU cores or the number

of GPUs. In Number of CPU cores/GPUS, enter a desired number.

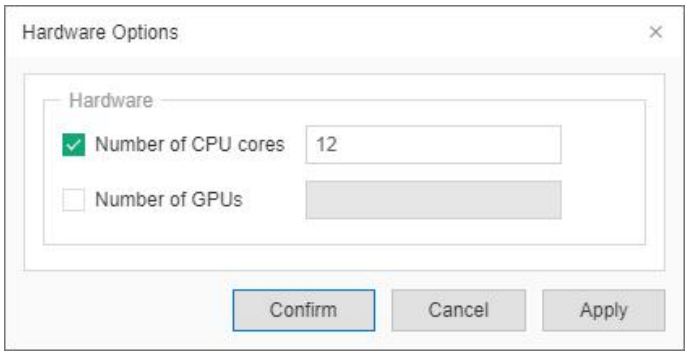


Figure 221 Hardware options

7.5.2 HPC

Click the **HPC**, select an algorithm, enter the number of nodes and the number of CPU cores. Click **Confirm**, submit the operation and upload the project to the server for simulation.

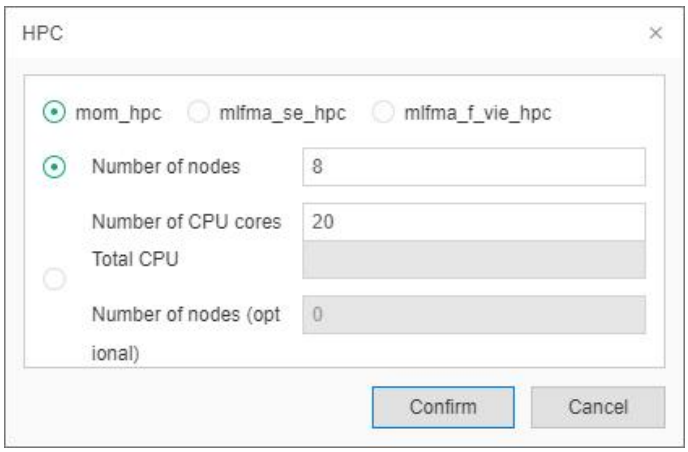


Figure 222 HPC

Note: Check whether the license supports before use. If the license does not support, an error will be reported.

7.5.3 Remote

The remote button configures the interface address and secret key that HPC (high-performance computing) uploads to the server.

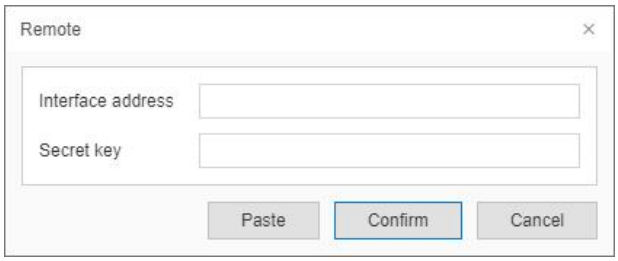


Figure 223 Remote

8 Result module

The Results module includes Import / Export, Results, Editing and Results Settings. Import and export include importing data, exporting data and pictures. Results include model parameters, currents, near field and far field results. Editing includes synthesis of results. Results settings include results export settings.

8.1 Import/Export

8.1.1 Import Data

Click Import Data to import external data in the selected result graph

8.1.2 Export Data

Click export data to export selected result map to a local path.

The steps to export data are:

1、 Select the result label that needs to be exported.

2、 Click the Export Data button.

3、 Select the curve to be exported.

4、 Choose a location to save file.

5、 Click **Save**.

8.1.3 Export Plot

Click export data to export selected result as a picture to a local path.

The steps to export plot are:

1、 Select the result label that needs to be exported.

2、 Click the Export Plot button.

3、 Choose a location to save file.

4、 Click **Save**.

8.2 Export Result Settings

Export result settings selects corresponding field source files to be exported.

8.2.1 Near Field

In Export result settings, click Near field to export the near field results as a near field source.enf file

The operation steps for setting and exporting near-field results are as follows:

1、 After the project finishes running, click the result setting export button - near field.

2、 In the near-field data window that pops up, check the near-field result settings that need to be exported.

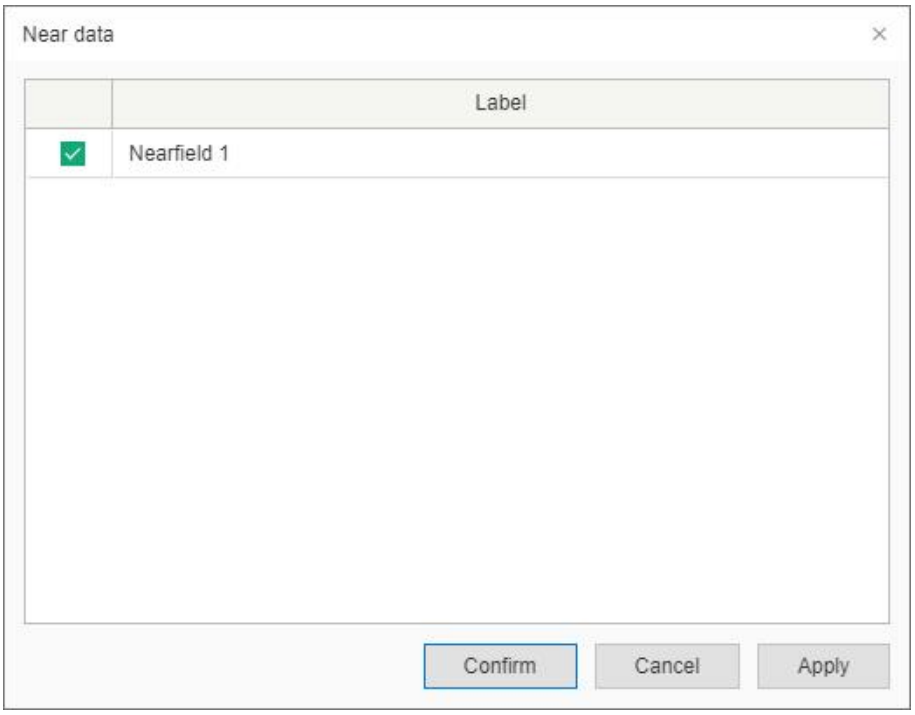


Figure 224 Near field export

3、 Click **Confirm**, select the export path, and click Save.

8.2.2 Far Field

In Export result settings, click Far-field to export the far field results as a far field.ff file and a far field CAE.ffp file.

The operation steps of setting and exporting the far-field results are as follows:

1、 After the project finishes running, click the result setting export button - far field, and choose to export far field data or far field port data.

2、 In the far-field data window, check the far-field result settings that need to be exported.

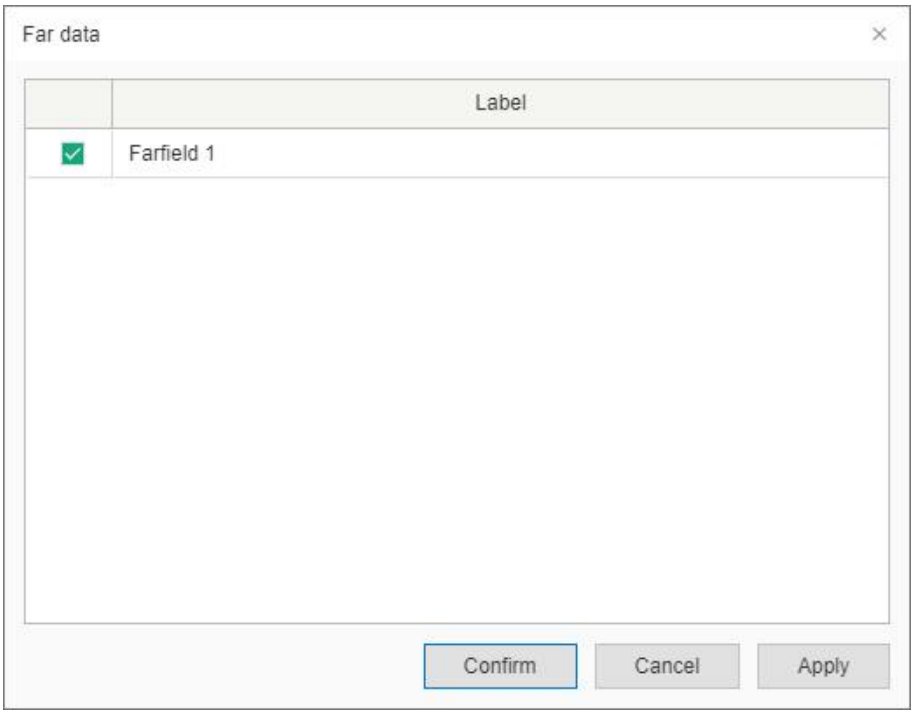


Figure 225 Far field export

3、 Click **Confirm**, select the export path, and click Save.

8.3 Result

Results include model parameters, currents, near fields and far fields.

8.3.1 Model parameters

Model parameters include S parameters, Y parameters, Z parameters, VSWR, Gamma and other parameters. Among the model parameters, users can draw 2D plots and Smith charts of these parameters.

8.3.1.1 2D plot

The steps of drawing 2D plots are as follows:

1、 Clic **Model Parameter** button .

2、 Next, click **2D plot** button in the drop-down box.

3、 Select the scan variable in the model parameter-2D plot window-scan variable.

4、 Select the corresponding label in sequence among mode, component, function and unit.

5、 Click **Draw** to draw a 2D plot.

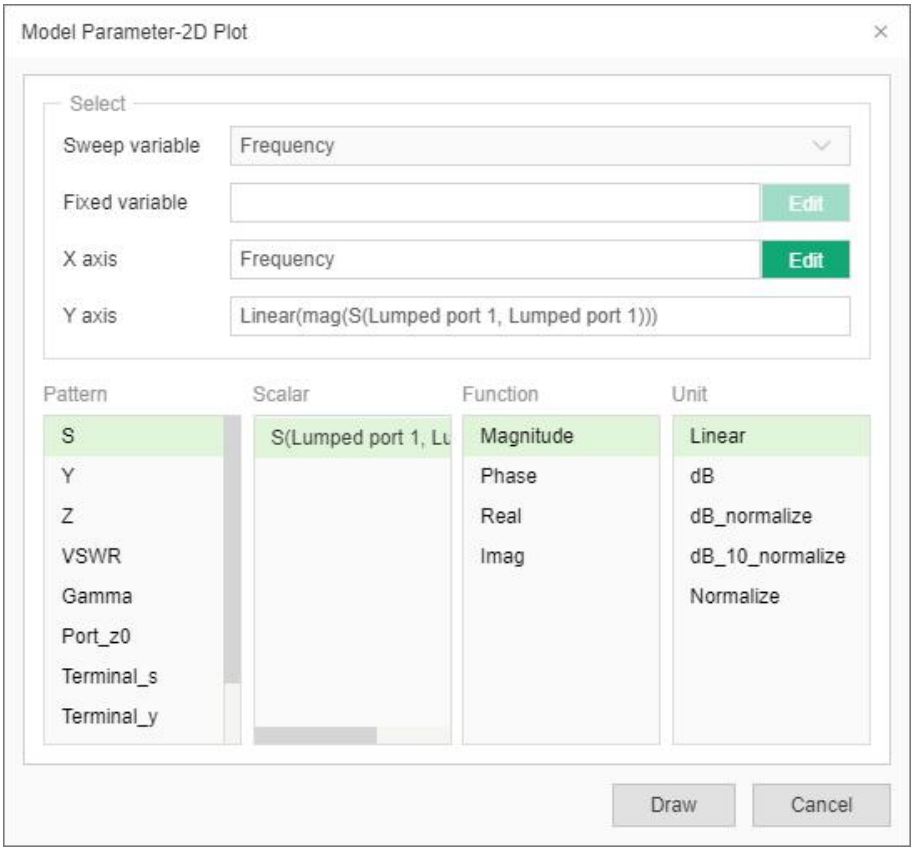


Figure 226 Model parameters — 2D Plots

8.3.1.2 Smith Chart

Smith chart are:

1、 Click **Model Parameter** button .

2、 Next, click **Smith chart** button.

3、 Select the scan variable in the model parameter-Smith chart window-scan variable.

4、 Select the corresponding label in sequence among mode, component, function and unit.

5、 Click **Draw** to draw a Smith Chart.

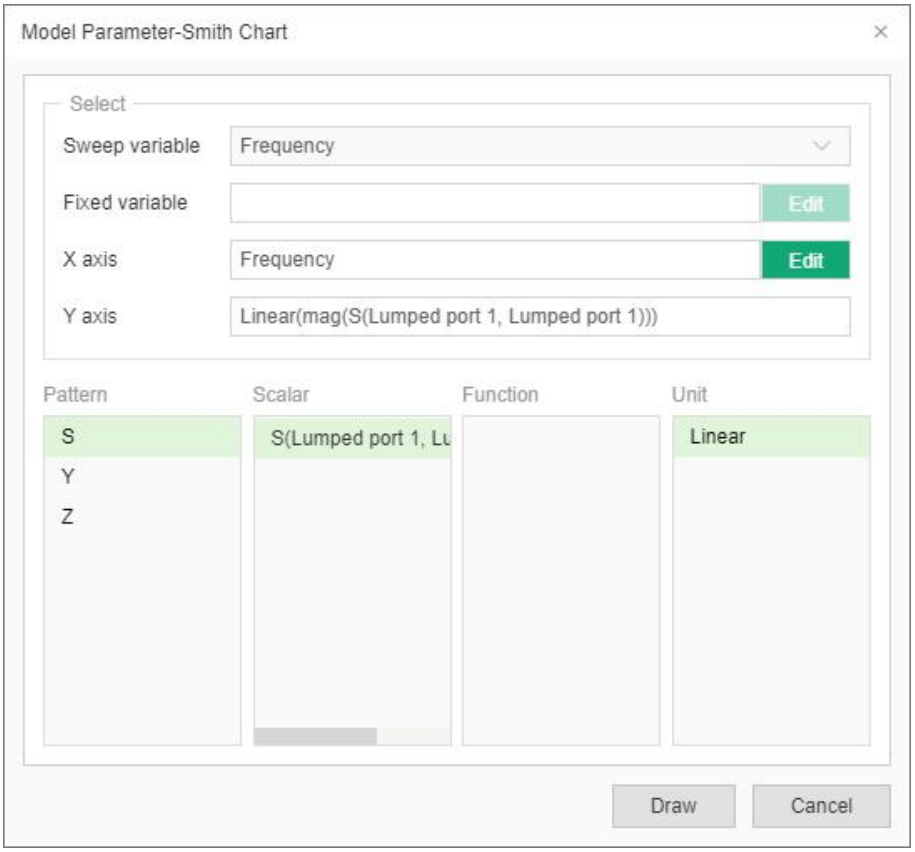


Figure 227 Model parameters— Smith Chart

8.3.2 Electric current

Click the current button to draw static current graph and dynamic current graph, including amplitude, instantaneous value and direction.

8.3.2.1 Static Current Diagram

The operation steps to draw the static current diagram are as follows:

1、 Click **Current** button .

2、 In the selection box of the current window, select the current result setting label

name and the excitation source label name accordingly.

3、 Next, select the values of frequency and other fixed variables in the input box.

4、 Select Mode, Components, and Units:

5、 Selection magnitude in the function menu.

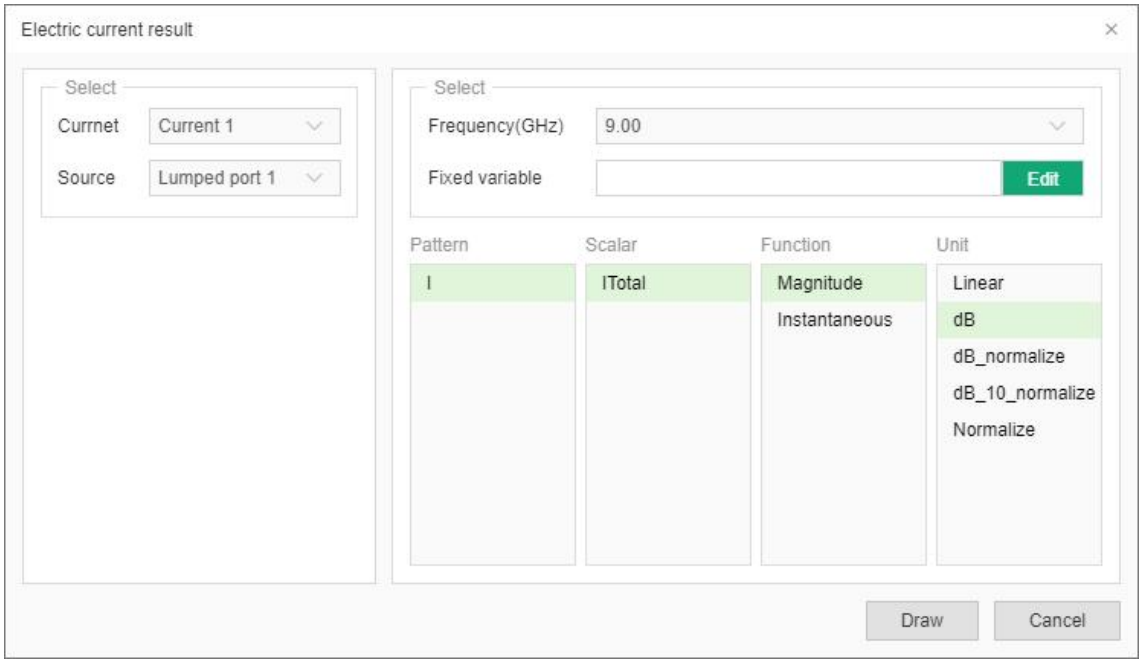


Figure 228 Static current

6、 Click **Draw** to draw a static current diagram.

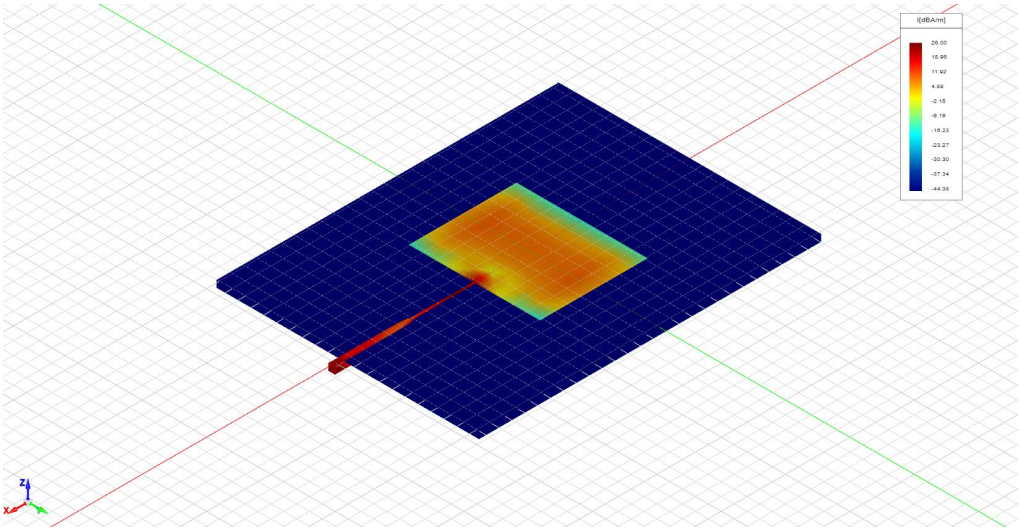


Figure 229 Static Current

7、 In the upper result view, you can adjust the viewing angle, zoom in/out the display of the result graph, perform all/single adaptive display of the viewing angle of

the result graph, and show/hide the display of the grid, stimulus and coordinate system.

8.3.2.2 Dynamic current diagram

The operation steps to draw the dynamic current diagram are as follows:

1、 Click **Current** button .

2、 In the selection box of the current window, select the current result setting label name and the excitation source label name accordingly.

3、 Next, select the values of frequency and other fixed variables in the input box.

4、 Select Mode, Components, and Units:

5、 Select instantaneous in function menu.

6、 Check the dynamic graph in the input box.

7、 Click **Draw** to draw a dynamic current diagram.

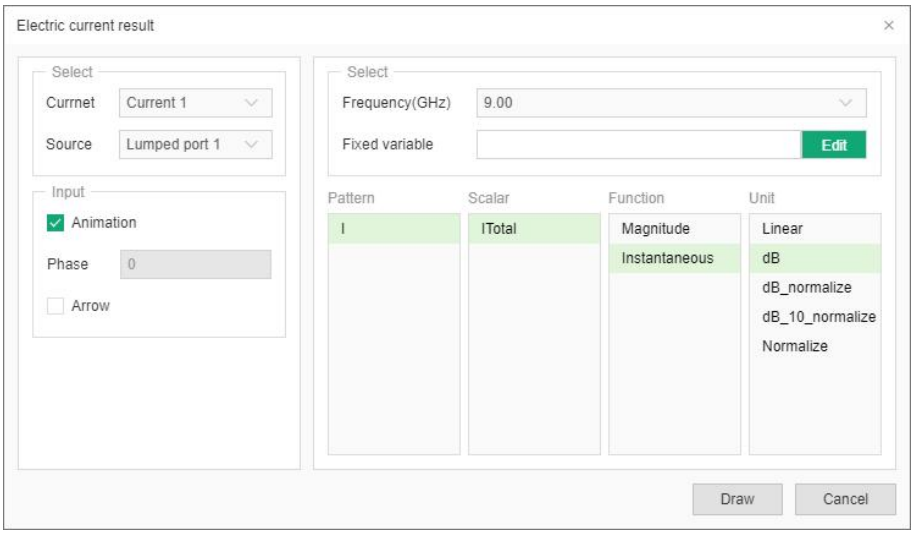


Figure 230 Dynamic Current

8、 In addition to adjusting the overall perspective of the dynamic current graph, the user can also pause and resume dynamic playback, set the speed and phase of the

dynamic graph change in the dynamic graph settings above.

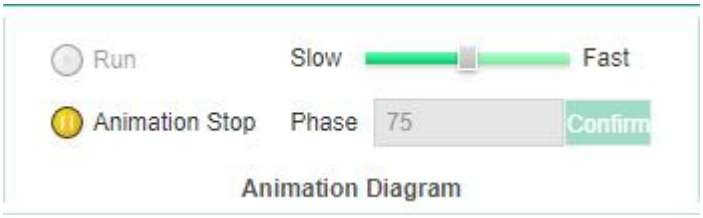


Figure 231 Dynamic image playback settings

8.3.3 Near field

In the near field results: electric fields, magnetic fields, cross-polarization resolution, gain patterns, etc. can be plotted. The result types includes 2D plot, 2D polar plot, 2D contour plot, 2D intensity plot, and 3D plot.

8.3.3.1 2D Plot

The steps of drawing 2D plots are as follows:

1、Click **Near Field** button.

2、Next, click **2D plot** button in the drop-down box.

3、In the selection box on the left side of the window of the 2D section view, select the near-field tag name and the excitation source tag name.

4、Select the scan variable in the scan variable.

5、Click the Edit button behind the fixed variable to enter the fixed variable window. In the window, click Edit behind each variable to select the value of the variable.

6、Select the corresponding label in sequence among mode, component, function and unit.

7、Click **Draw** to draw a 2D section view.

8、After drawing the result graph, you can choose to add Mark points, calculate, import/export data, etc. in the result graph interface.

9、Right-click and select Properties to set the properties of the current result plot.

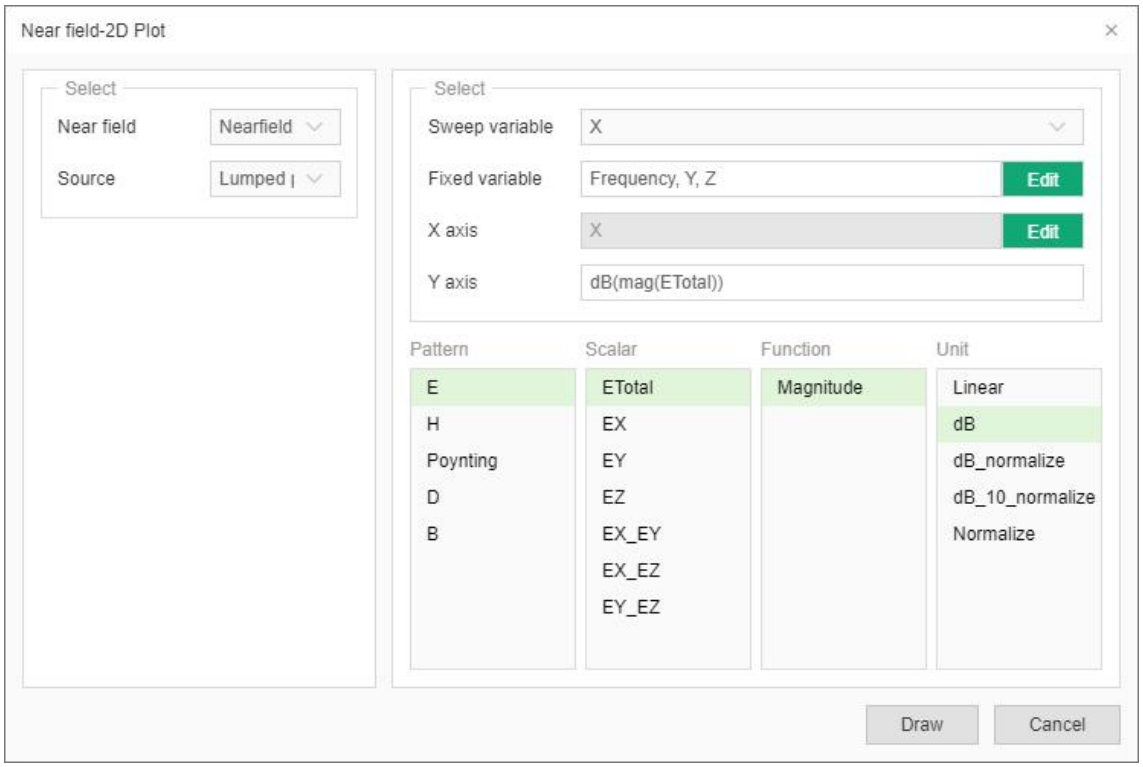


Figure 232 Far Field - 2D Plot

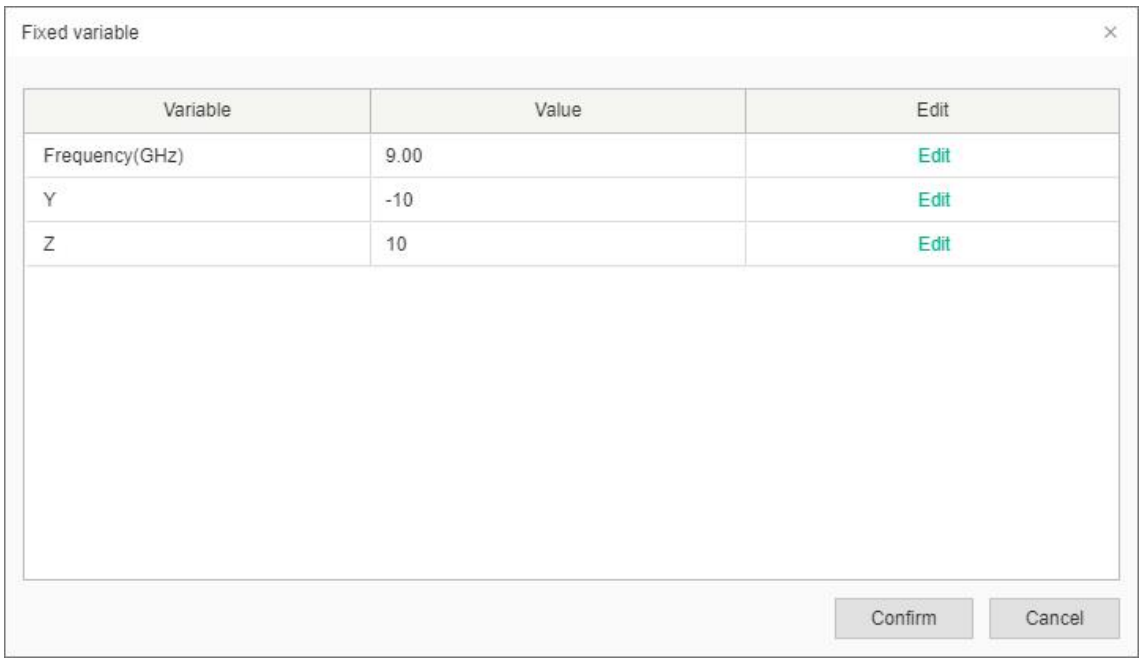


Figure 233 Near field - fixed variable

10、At this time, the near-field tag name will appear under the project manager area-result tree. Click the sub-tab under this tag to modify the near -field 2D plot.

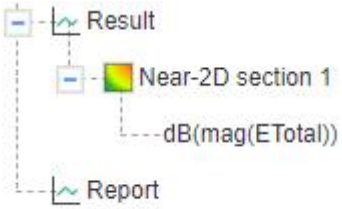


Figure 234 Near field - 2D plot

8.3.3.2 2D Contour Plot

The steps to draw a 2D contour plot are:

1、Click **Near Field** button.

2、Next, click **2D contour plot** button in the drop-down box.

3、Select the far-field tag name and excitation source tag name in the left selection box of the 2D contour plot window.

4、Select the scan variable in the scan variable.

5、Click the Edit button behind the fixed variable to enter the fixed variable window. In the window, click Edit behind each variable to select the value of the variable.

6、Select the variables of the X-axis and Y-axis (the variables of the X-axis and Y-axis can only select the variables in the scanning variables).

7、Select the corresponding label in sequence among mode, component, function and unit.

8、Click **Draw** to draw a 2D contour plot.

9、After drawing the result graph, you can choose to import/export data and other operations in the result graph interface.

10、Right-click on the result graph interface and select Properties to set the properties of the current result graph.

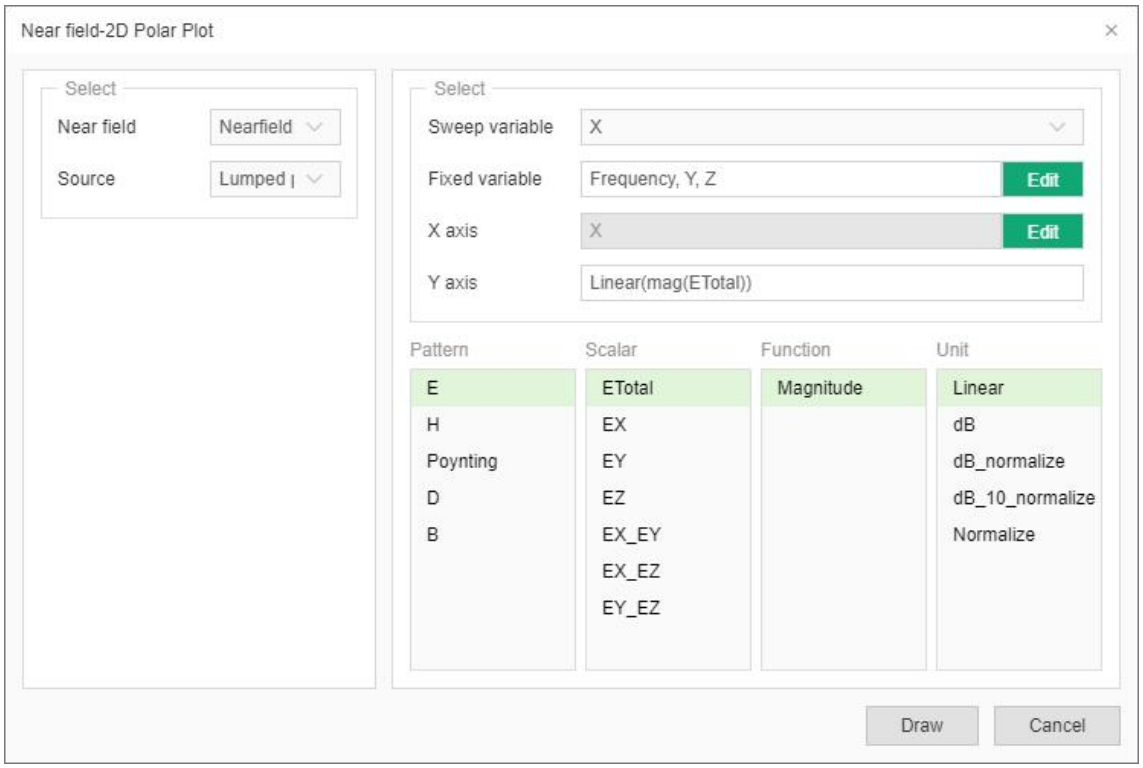


Figure 235 Near Field - 2D Contour plot

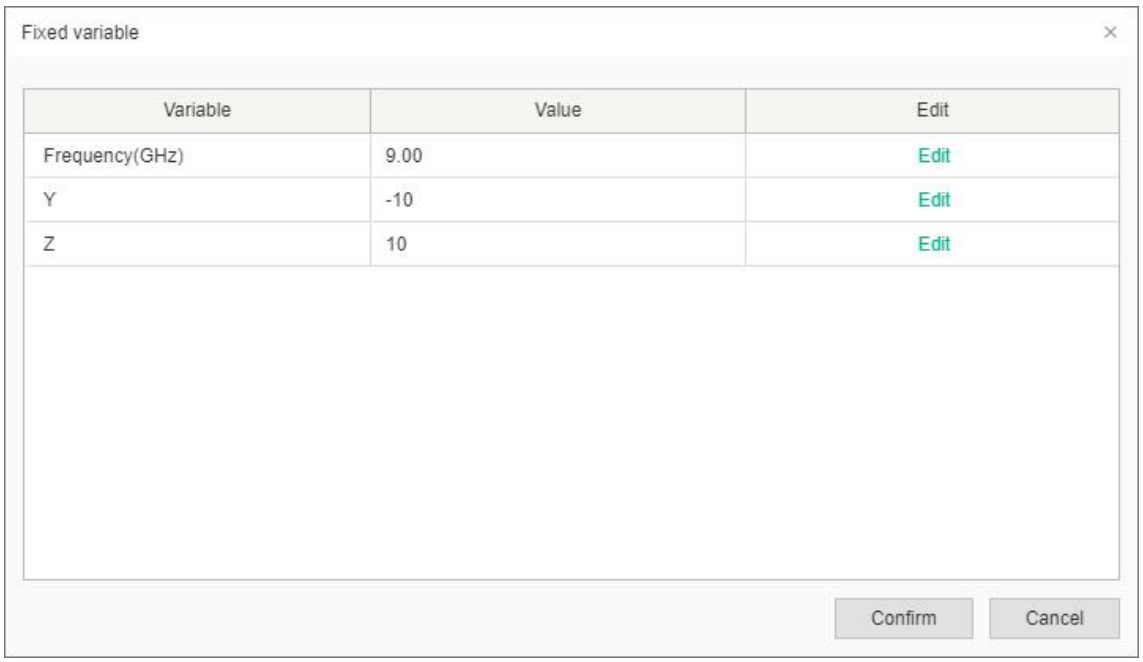


Figure 236 Near field - fixed variable

11、At this time, the far-field label name will appear under the project manager area-result tree. Click the sub-label under this label to modify the near-field 2D contour plot.

Figure 237 Near Field - 2D Contour plot

8.3.3.3 2D Intensity Plot

The steps to draw a 2D intensity plot are:

1、Click **Near Field** button.

2、Next, click **2D Intensity plot** button in the drop-down box.

3、In the selection box on the left side of the 2D intensity plot window, select the near-field tag name and the excitation source tag name.

4、Select the scan variable in the scan variable.

5、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

6、Select the variables of the X-axis and Y-axis (the variables of the X-axis and Y-axis can only select the variables in the scanning variables).

7、Select the corresponding label in sequence among mode, component, function and unit.

8、Click Draw to draw a 2D intensity plot.

9、After drawing the result graph, you can choose to import/export data and other operations in the result graph interface.

10、Right-click on the result graph interface and select Properties to set the properties of the current result graph.

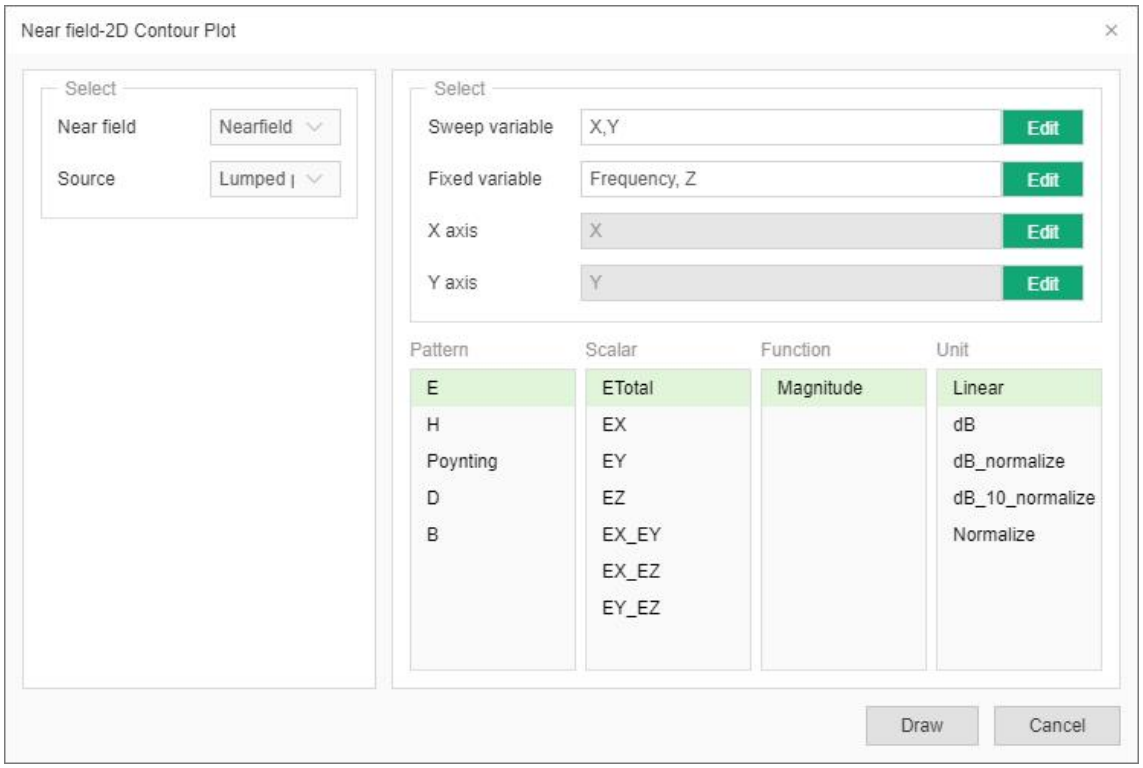


Figure 238 Near-field - 2D Intensity plot

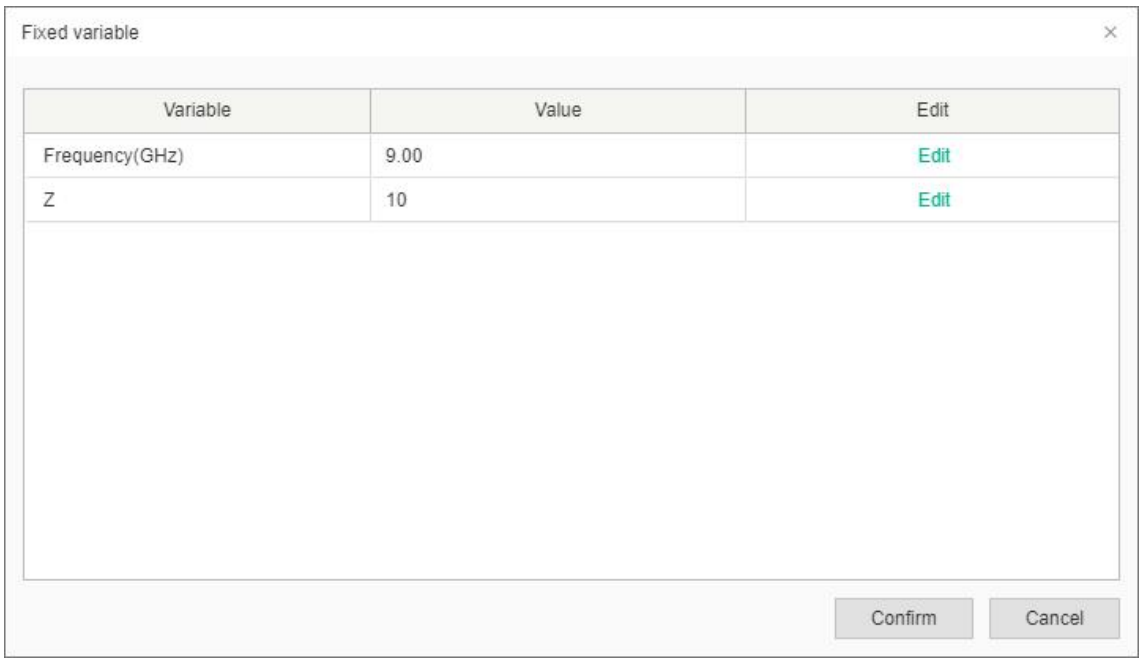


Figure 239 Near field - fixed variable

11、At this time, the far-field tag name will appear under the project manager area-result tree. Click the sub-tab under the tag to modify the far-field 2D intensity plot.

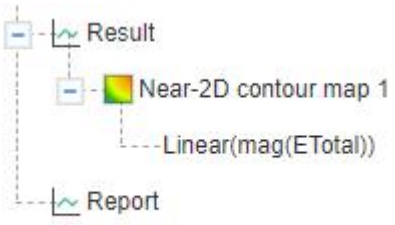


Figure 240 Near Field - 2D Intensity plot

8.3.3.4 3D Plot

The steps of drawing 3D Cartesian coordinate system plots are:

1、 Click **Near Field** button.

2、 Next, click **3D Plot** button in the drop-down box.

3、 In the selection box on the left side of the 3D Cartesian coordinate system window, select the near-field tag name and the excitation tag name.

4、 Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

5、 Select the corresponding label in sequence among mode, component, function and unit.

6、 Click **Draw** to draw a 3D Cartesian coordinate system result plot.

7、 Right-click on the result graph interface and select Properties to set the properties of the current result graph.

8、 In the upper box, you can adjust the viewing angle, zoom in/out the display of the result graph, perform all/single adaptive display of the viewing angle of the result graph, and display/hide the display of the grid, stimulus and coordinate system.

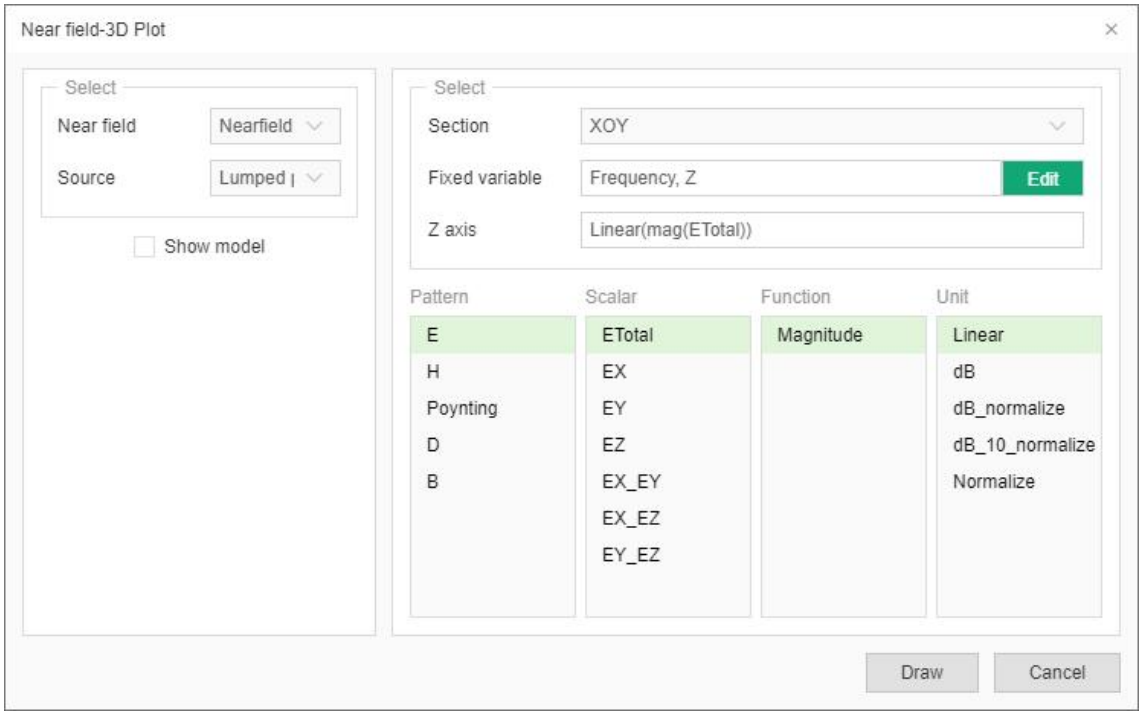


Figure 241 Near-field — 3D Plot

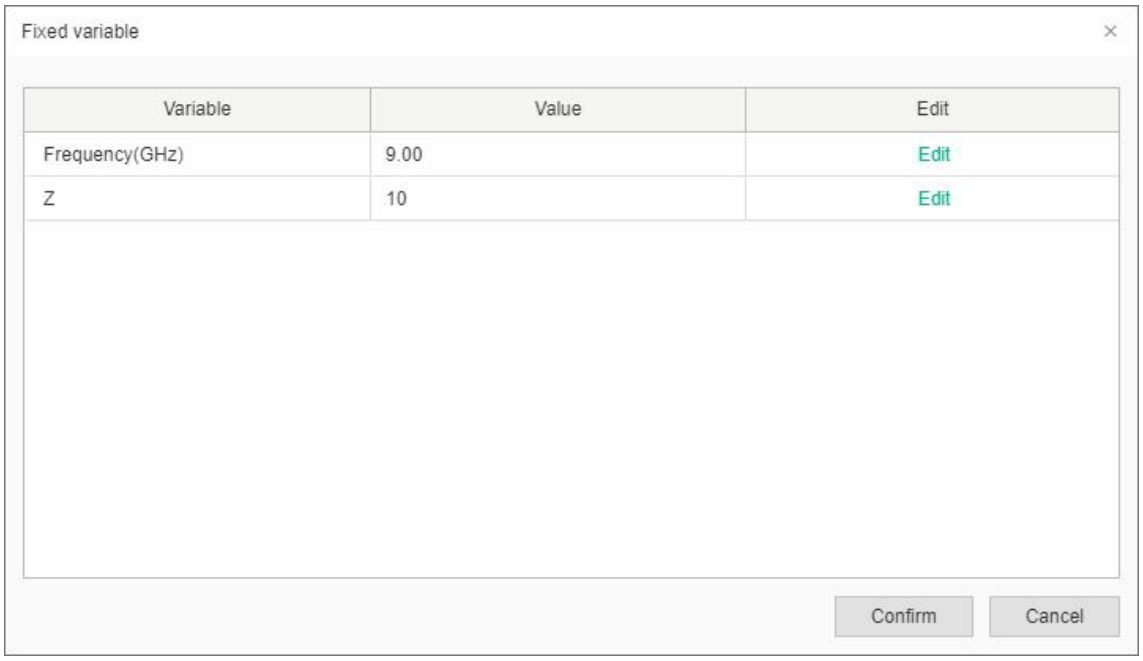


Figure 242 Near field - fixed variable

9、 At this time, the near-field tag name will appear under the project manager area-result tree. Click the sub-tab under this tag to modify the near-field 3D Cartesian coordinate system plot.

Figure 243 Near Field - 3D Plot

8.3.4 Far field

In the far field results: electric field, directivity coefficient, axial ratio, cross-polarization resolution, gain pattern, etc. can be plotted. The result types include 2D slice plots, 2D polar plots, 2D contour plots, 2D intensity plots, 3D Cartesian coordinates, 3D cylindrical coordinates, and 3D spherical coordinates.

8.3.4.1 2D Plot

The steps of drawing a 2D plot are as follows:

1、Click the **Far Field** button .

2、Next, click **2D Plot** button in the drop-down box.

3、In the selection box on the left side of the window of the 2D section view, select the far-field tag name and the excitation source tag name.

4、Select the scan variable in the scan variable.

5、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

6、Select the corresponding label in sequence among mode, component, function and unit.

7、Click **Draw** to draw a 2D section view.

8、After drawing the result graph, you can choose to add Mark points, calculate, import/export data, etc. in the result graph interface.

9、Right-click and select Properties to set the properties of the current result plot.

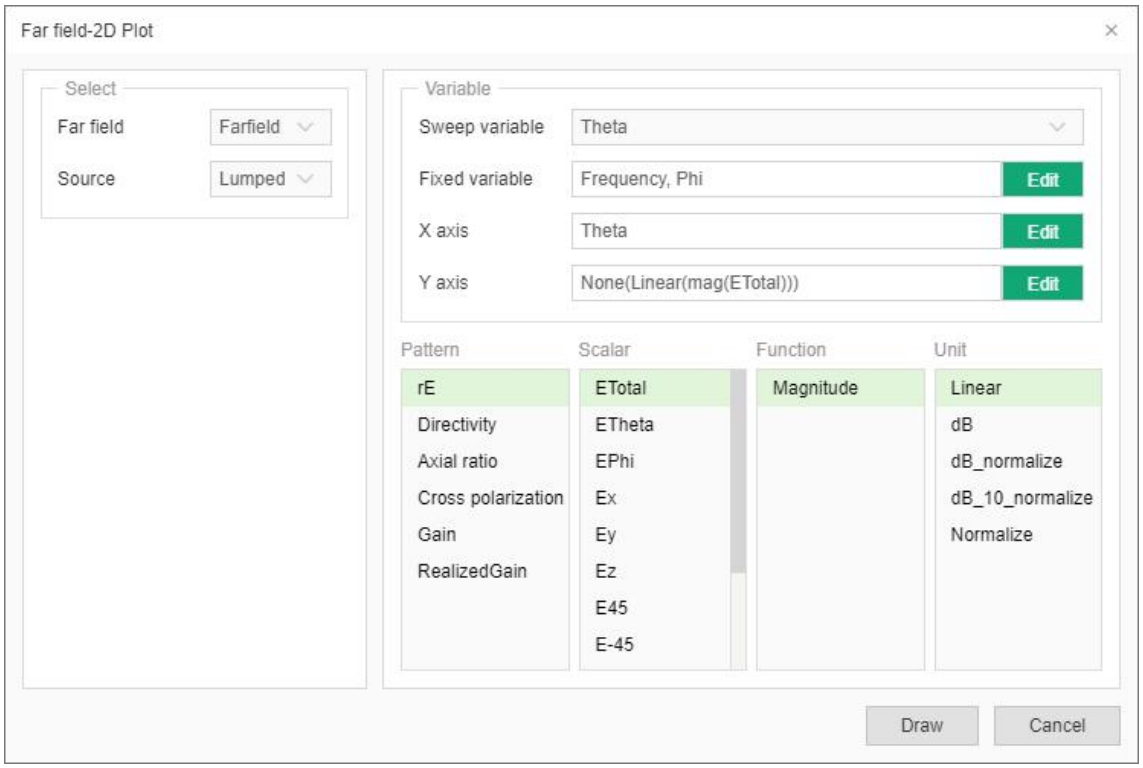


Figure 244 Far-field - 2D Plot

Figure 245 Far-field - fixed variable

10、At this time, the far-field tag name will appear under the project manager area-result tree. Click the sub-tab under this tag to modify the far-field 2D plot.

Figure 246 Far field - 2D section

8.3.4.2 2D Polar Plot

The steps to draw a 2D polar plot are:

1、 Click the **Far Field** button .

2、 Next, click **2D polar** button in the drop-down box.

3、 In the selection box on the left side of the 2D polar coordinate window, select the far-field tag name and the excitation source tag name.

4、 Select the scan variable in the scan variable.

5、 Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

6、 Select the corresponding label in sequence among mode, component, function and unit.

7、 Click **Draw** to draw a 2D polar plot.

8、 After drawing the result graph, you can choose to add Mark points, import/export data, etc. in the result graph interface.

9、 Right-click and select Properties to set the properties of the current result plot.

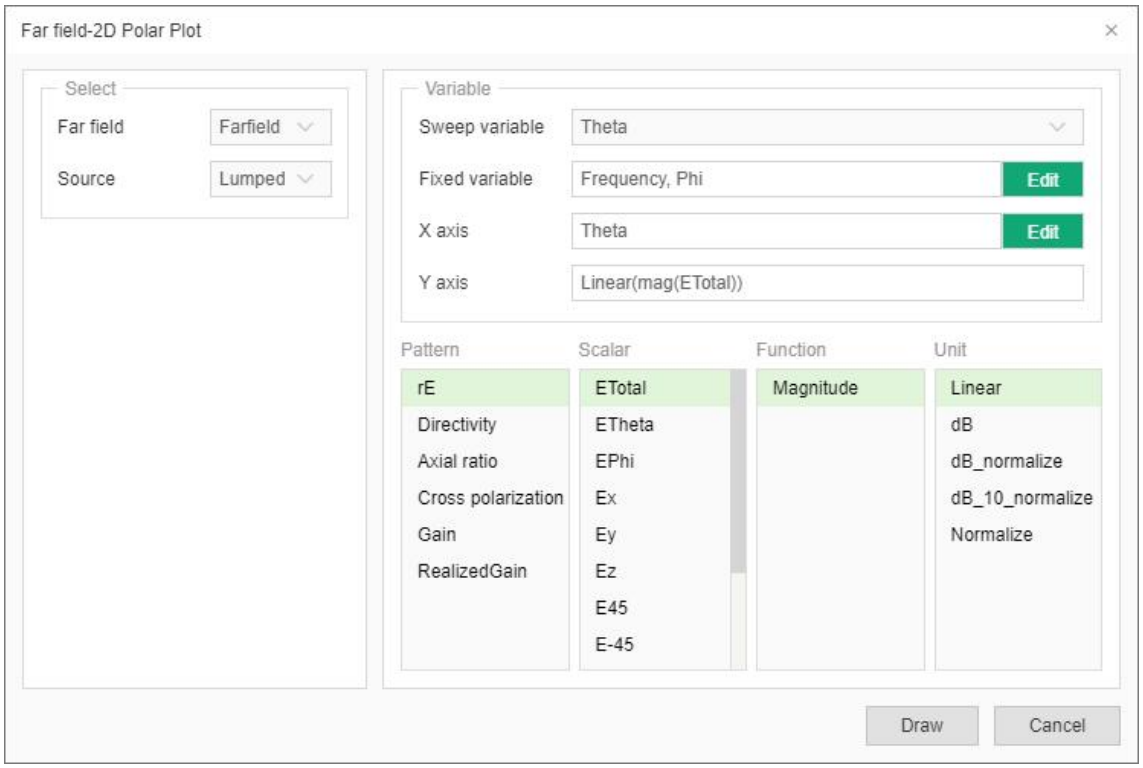


Figure 247 Far Field - 2D Polar Plot

Figure 248 Far field - fixed variable

10、 At this time, the far-field label name will appear under the project manager area-result tree. Click the sub-label under this label to modify the far-field 2D polar coordinate plot.

Figure 249 Far Field - 2D Polar Plot

8.3.4.3 2D Contour Plot

The steps to draw a 2D contour plot are:

1、Click the **Far Field** button .

2、Next, click **2D contour plot** button in the drop-down box.

3、Select the far-field tag name and excitation source tag name in the left selection box of the 2D contour plot window.

4、Select the scan variable in the scan variable.

5、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

6、Select the variables of the X-axis and Y-axis (the variables of the X-axis and Y-axis can only select the variables in the scanning variables).

7、Select the corresponding label in sequence among mode, component, function and unit.

8、Click **Draw** to draw a 2D contour plot.

9、After drawing the result graph, you can choose to import/export data and other operations in the result graph interface.

10、Right-click and select Properties to set the properties of the current result plot.

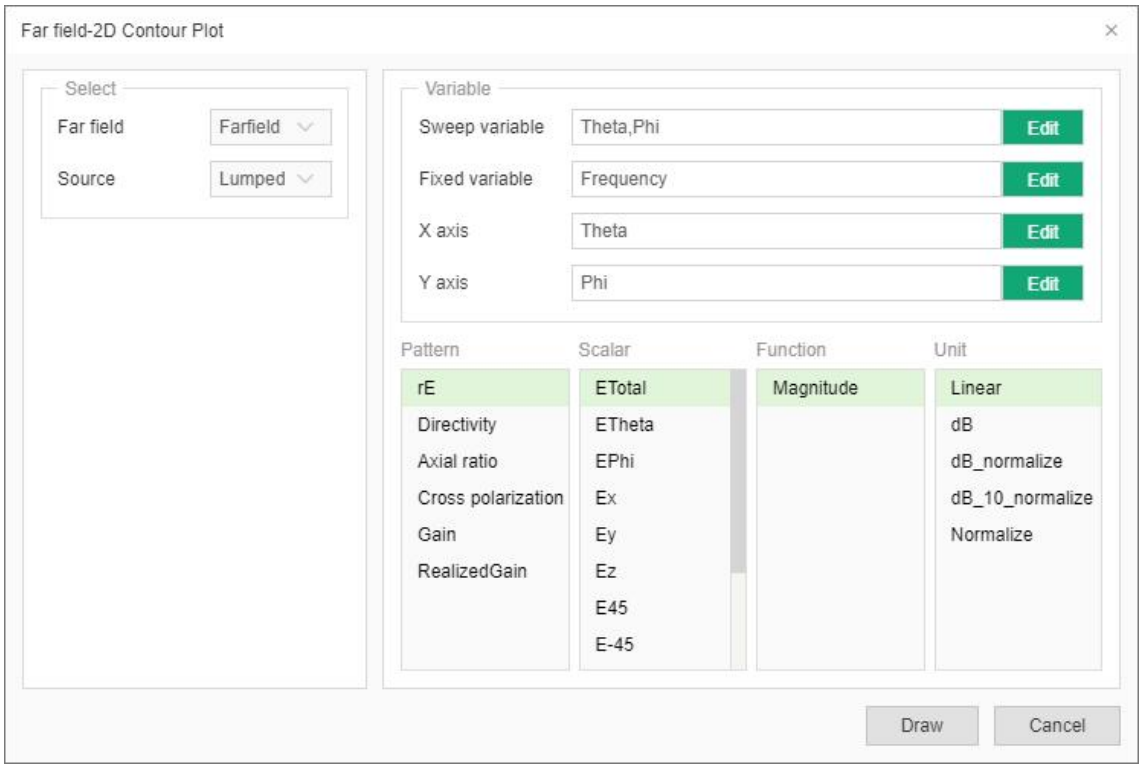


Figure 250 Far Field - 2D Contour plot

Figure 251 Far field - fixed variable

11、At this time, the far-field tag name will appear under the project manager area-result tree. Click the sub-tab under the tag to modify the far-field 2D contour plot.

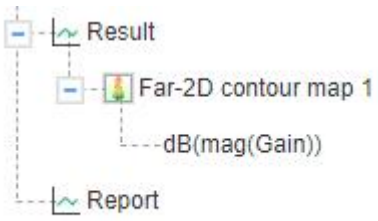


Figure 252 Far Field - 2D Contour plot

8.3.4.4 2D Intensity Plot

The steps to draw a 2D intensity plot are:

1、Click the **Far Field** button .

2、Next, click **2D Intensity plot** button in the drop-down box.

3、In the selection box on the left side of the 2D intensity plot window, select the far-field tag name and the excitation source tag name.

4、Select the scan variable in the scan variable.

5、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

6、Select the variables of the X-axis and Y-axis (the variables of the X-axis and Y-axis can only select the variables in the scanning variables).

7、Select the corresponding label in sequence among mode, component, function and unit.

8、Click **Draw** to draw a 2D intensity plot.

9、After drawing the result graph, you can choose to import/export data and other operations in the result graph interface.

10、Right-click and select Properties to set the properties of the current result plot.

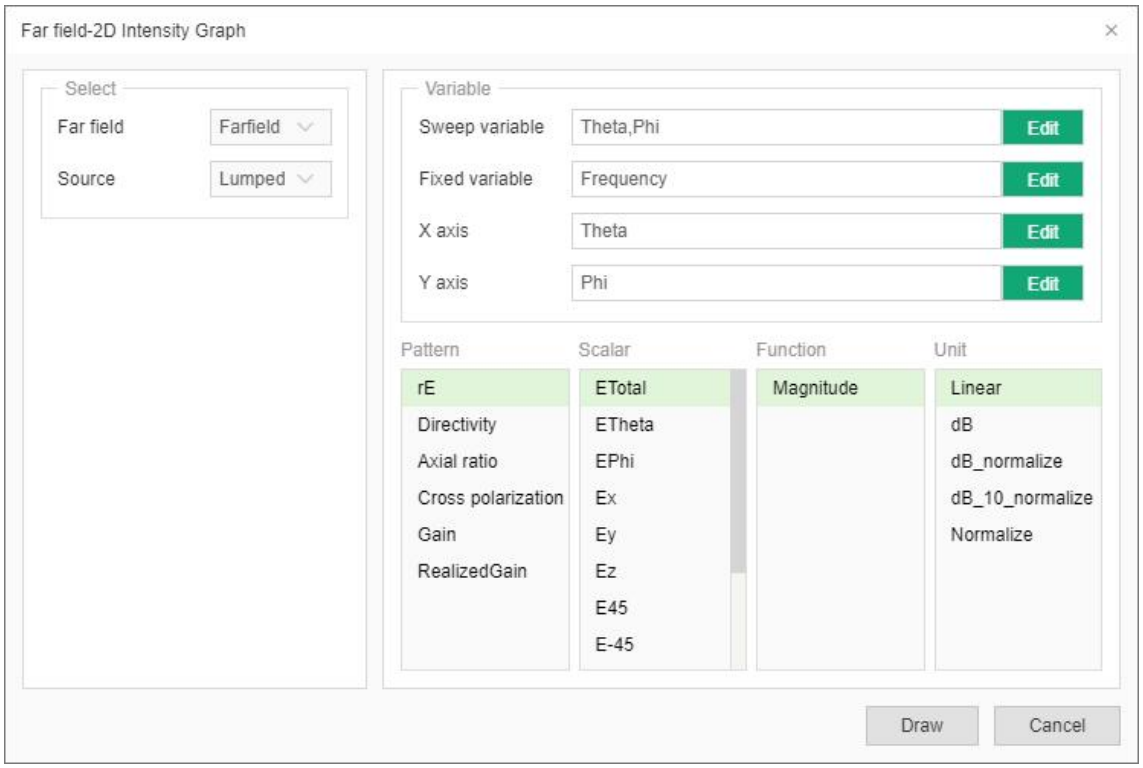


Figure 253 Far Field - 2D Intensity plot

Figure 254 Far field - fixed variable

11、At this time, the far-field result icon signature will appear under the project manager area-result tree. Click the sub-tab under this tab to modify the far-field 2D intensity plot.

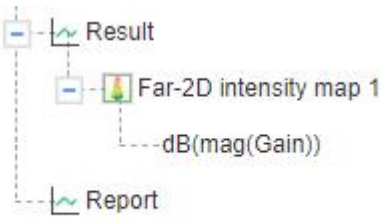


Figure 255 Far Field - 2D Intensity plot

8.3.4.5 3D Rectangular

The steps of drawing a 3D Rectangular are:

1、Click the **Far Field** button .

2、Next, click **3D Rectangular** button in the drop-down box.

3、In the selection box on the left side of the 3D Cartesian coordinate system window, select the far-field tag name and the excitation tag name.

4、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

5、Select the corresponding label in sequence among mode, component, function and unit.

6、Check whether to display the model according to your needs

7、Click **Draw** to draw a 3D Cartesian coordinate system.

8、Right-click and select Properties to set the properties of the current result plot.

9、In the upper box, you can adjust the viewing angle, zoom in/out the display of the result graph, perform all/single adaptive display of the viewing angle of the result graph, and display/hide the display of the grid, stimulus and coordinate system.

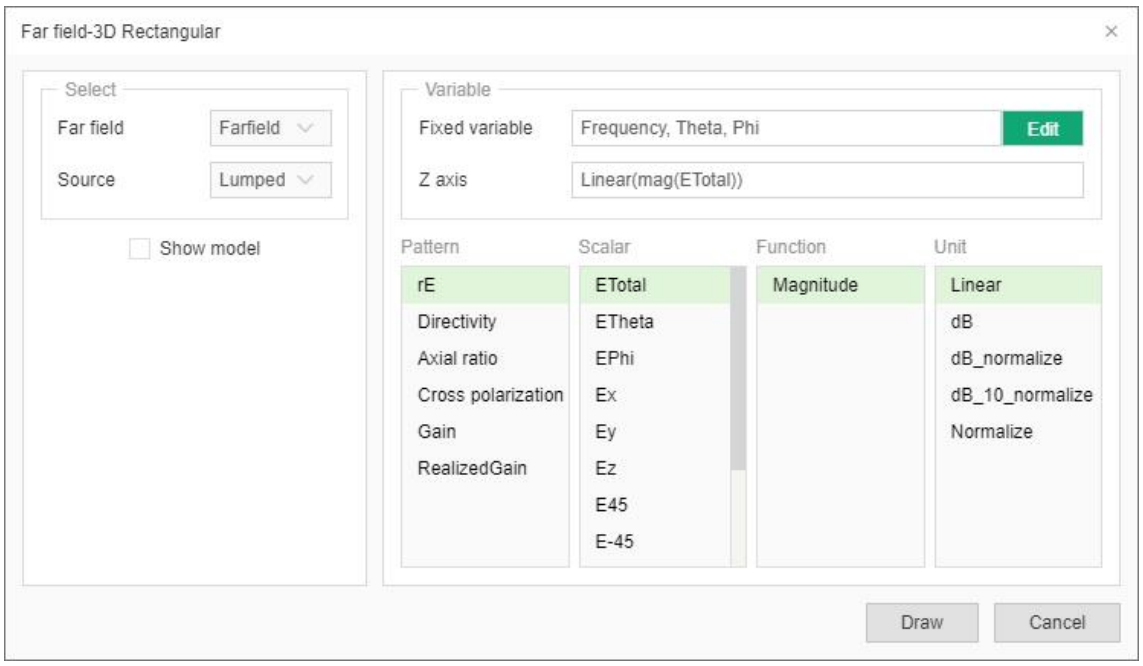


Figure 256 Far Field — 3D Rectangular

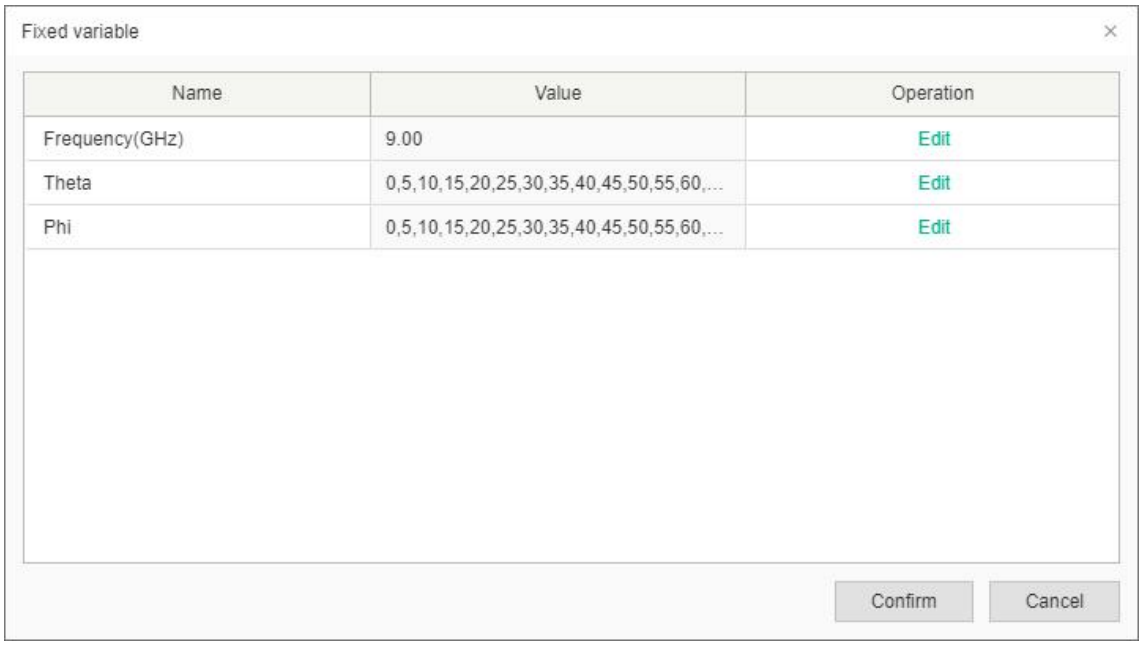


Figure 257 Far field - fixed variable

10、At this time, the far-field tag name will appear under the project manager area-result tree. Click the sub-tab under this tag to modify the far-field 3D Cartesian coordinate system plot.

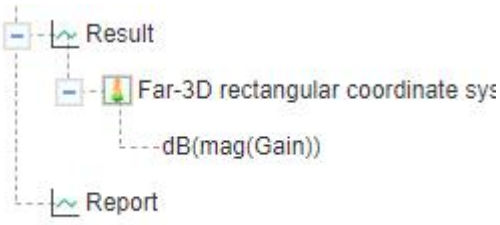


Figure 258 Far Field - 3D Rectangular

8.3.4.6 3D Cylindrical

The steps of drawing a 3D cylindrical plot are:

1、Click the **Far Field** button .

2、Next, click **3D Cylindrical** button in the drop-down box.

3、In the selection box on the left side of the 3D cylindrical coordinate system window, select the far-field tag name and the excitation tag name.

4、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

5、Select the corresponding label in sequence among mode, component, function and unit.

6、Check whether to display the model according to your needs

7、Click **Draw** to draw a 3D cylindrical coordinate system.

8、Right-click and select Properties to set the properties of the current result plot.

9、In the upper box, you can adjust the viewing angle, zoom in/out the display of the result graph, perform all/single adaptive display of the viewing angle of the result graph, and display/hide the display of the grid, stimulus and coordinate system.

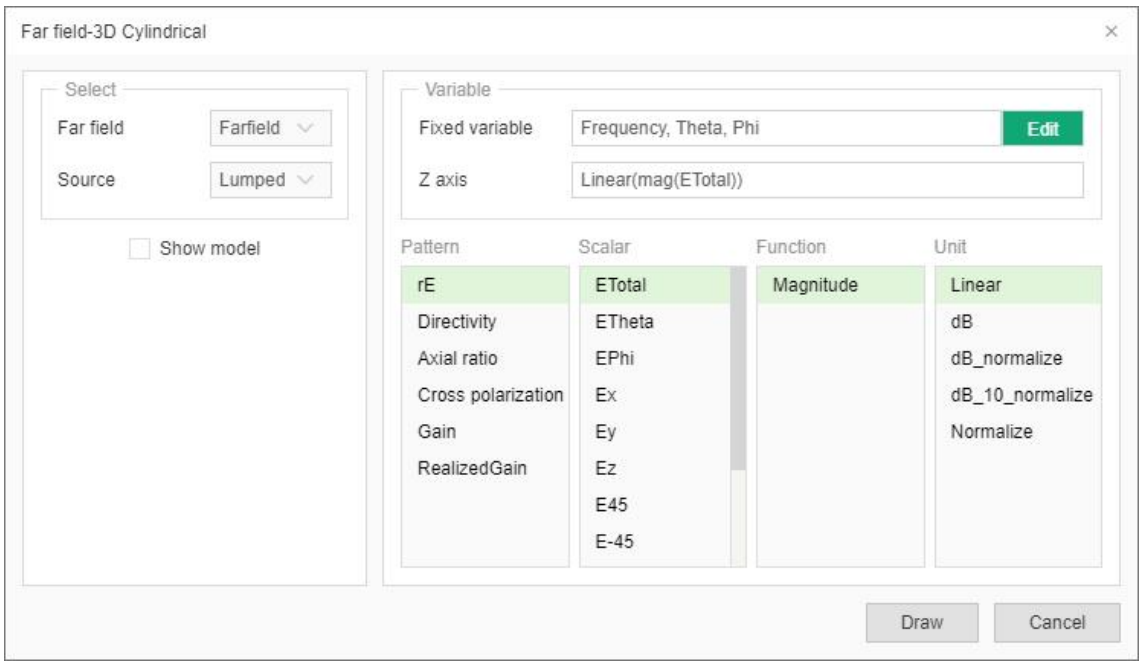


Figure 259 Far Field — 3D Cylindrical

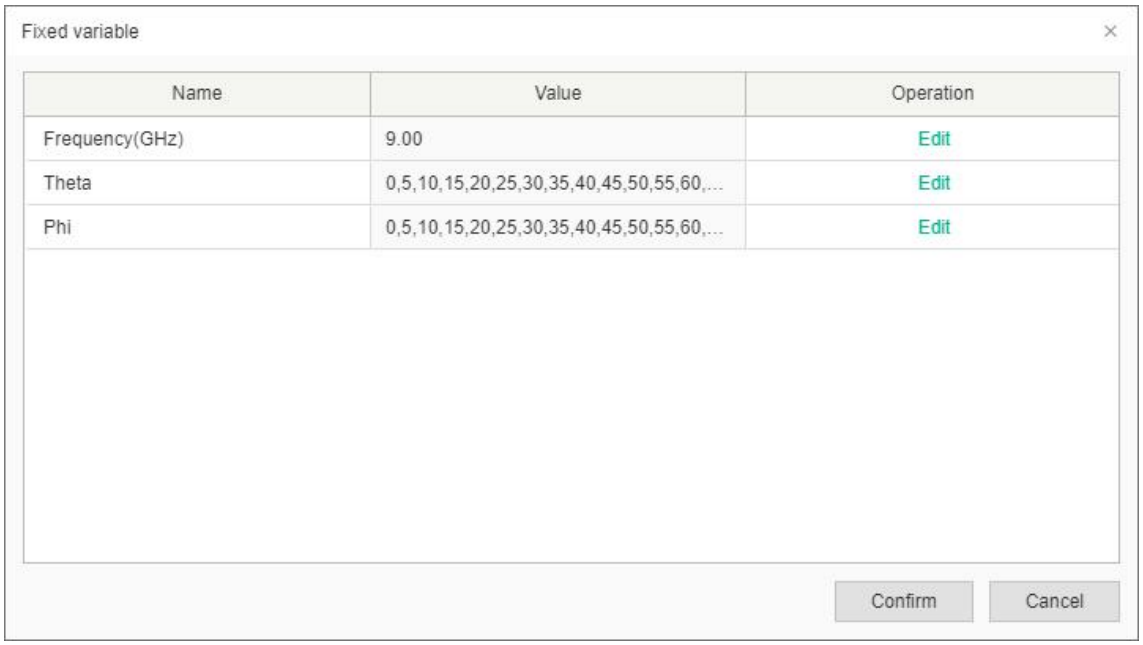


Figure 260 Far field - fixed variable

10、At this time, the far-field tag name will appear under the project manager area-result tree. Click the sub-tab under this tag to modify the far-field 3D cylindrical coordinate system plot.

Figure 261 Far Field - 3D Cylindrical

8.3.4.7 3D Spherical

The steps of drawing a 3D spherical plot are:

1、Click the **Far Field** button .

2、Next, click **3D Spherical** button in the drop-down box.

3、In the selection box on the left side of the 3D spherical coordinate system window, select the far-field tag name and the excitation tag name.

4、Click the Edit button behind the fixed variable to enter the fixed variable window. In the pop-up window, click Edit behind each variable to select the value of the variable.

5、Select the corresponding label in sequence among mode, component, function and unit.

6、Check whether to display the model according to your needs

7、Click **Draw** to draw a 3D spherical coordinate system.

8、Right-click and select Properties to set the properties of the current result plot.

9、In the upper box, you can adjust the viewing angle, zoom in/out the display of the result graph, perform all/single adaptive display of the viewing angle of the result graph, and display/hide the display of the grid, stimulus and coordinate system.

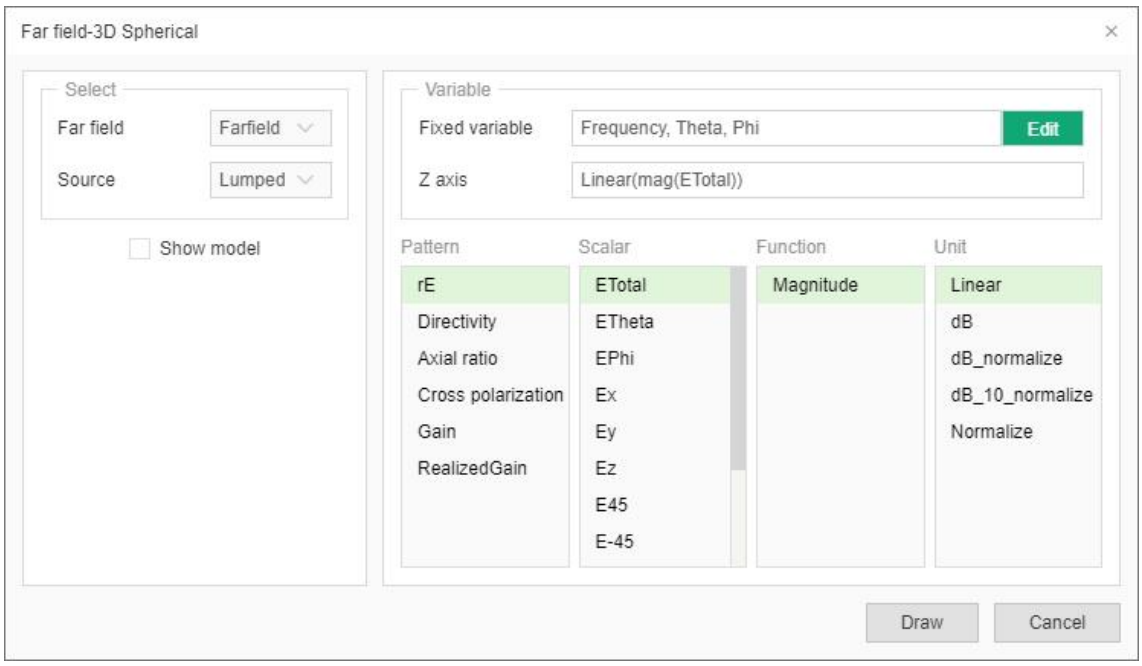


Figure 262 Far Field — 3D Spherical

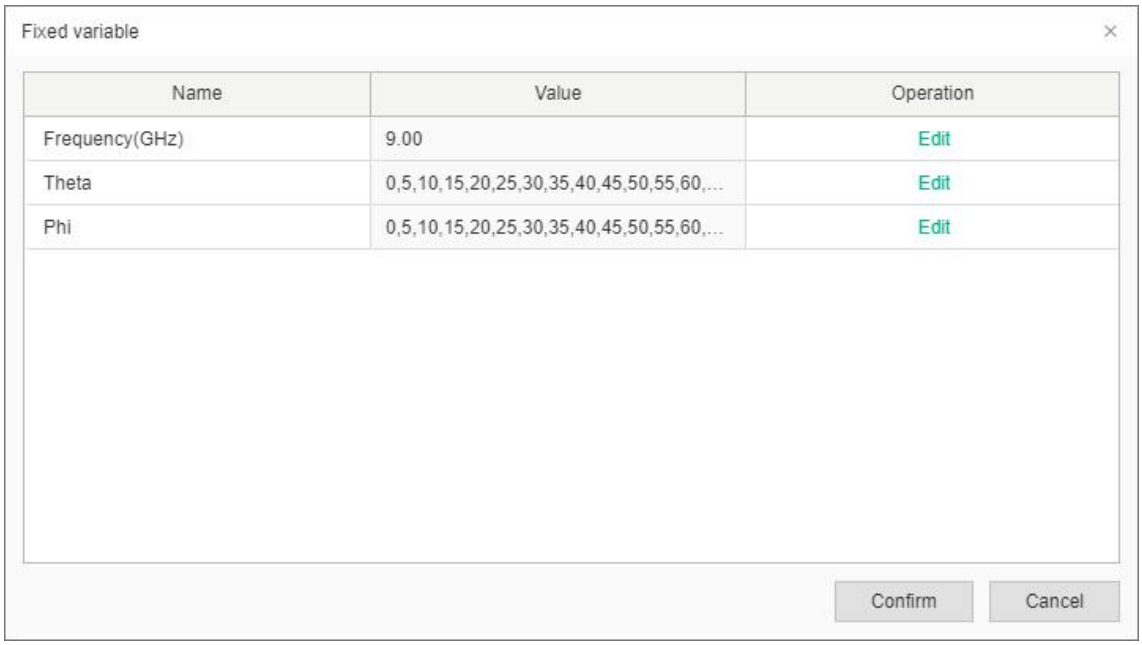


Figure 263 Far field - fixed variable

10、At this time, the far-field tag name will appear under the project manager area-result tree. Click the sub-tab under this tag to modify the far-field 3D spherical coordinate system plot.

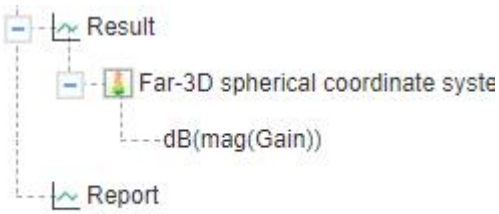


Figure 264 Far Field - 3D Spherical