Altair

HyperWorks®

Getting Started Manual for FEKO 2017.2

Getting Started Guide

2017.2 RELEASE

July 2017

Altair® HyperWorks® Version 2017.2
A Platform for Innovation®

Altair Engineering Support Contact Information

Web site	http://www.altairhyperworks.com http://www.altairhyperworks.com/ClientCenterHWSupportProduct.aspx http://www.altairhyperworks.com/feko

Altair® HyperWorks® v. 2017.2
A Platform for Innovation™

HyperWorks® products:

HyperMesh® ©1990–2017; HyperCrash® ©2001–2017; OptiStruct® ©1996–2017; RADIOSS® ©1986–2017; HyperView® ©1999–2017; HyperView Player® ©2001–2017; HyperMath® ©2007-2017; HyperStudy® ©1999–2017; HyperGraph® ©1995–2017; MotionView® ©1993–2017; MotionSolve® ©2002–2017; HyperForm® ©1998–2017; HyperXtrude® ©1999–2017; Process Manager™ ©2003–2017; Templex™ ©1990-2017; TextView™ ©1996-2017; MediaView™ ©1999-2017; TableView™ ©2013-2017; BatchMesher™©2003–2017; HyperWeld® ©2009–2017; HyperMold® ©2009–2017; Manufacturing Solutions™ ©2005–2017; solidThinking Inspire® ©2017 2009–2017; solidThinking Evolve® 2017 ©1993–2017; Durability Director™ ©2009–2017; Suspension Director™ ©2009–2017; AcuSolve® ©1997–2017; AcuConsole® ©2006–2017; SimLab® ©2004–2017; Virtual Wind Tunnel™ ©2012–2017; FEKO® (©1999-2014 Altair Development S.A. (Pty) Ltd.; ©2014-2017 Altair Engineering, Inc.); ConnectMe™ ©2014–2017; Click2Extrude™ Polymer 2017 ©1996-2017; Click2Extrude™ Metal 2017 ©1996-2017; Click2Form™ 2017 ©1998-2017.

Additional Altair Products:
Multiscale Designer™ ©2011–2017;
Flux v.12.2 ©1983–2017;
InCa3D v.3.1 ©1996–2016;
CDE v.2 ©2012–2016;
Got-It v.3 ©2002–2016;
WinProp v.14.5 ©2000–2017

Altair Packaged Solution Offerings (PSOs):
Automated Reporting Director™ ©2008–2017; GeoMechanics Director ©2011–2017; Impact Simulation Director™ ©2010–2017; Model Mesher Director™ ©2010–2017; Model Verification Director™ ©2013–2017; NVH Director™ ©2010–2017; Squeak and Rattle Director™ ©2012–2017; Virtual Gauge Director™ ©2012–2017; Weight Analytics™ ©2013–2017; Weld Certification Director™ ©2014–2017; Multi-Disciplinary Optimization™ ©2012-2017.

Altair Simulation Cloud Suite:

Simulation Manager™ ©2003–2017; Compute Manager™ ©2003–2017; Display Manager™ ©2003–2017; and Process Manager™ ©2003–2017.

Third party software licenses

Software Security Measures:

Contents

1 Getting started project: A rectangular horn

1.1 Example overview

This example uses a completed model to familiarise the user with the FEKO components and workflow. It is intended for users with little or no experience with Altair FEKO. The various aspects of the different FEKO components are discussed as they are encountered. A demonstration model of a completed horn antenna with results is used as an example. The model will not be constructed as part of this demonstration, but instead be used to demonstrate tasks that are often required in the FEKO components.

The horn model is similar to the model used in the demo video[1]. It is recommended that the demo video be watched before working through this example.

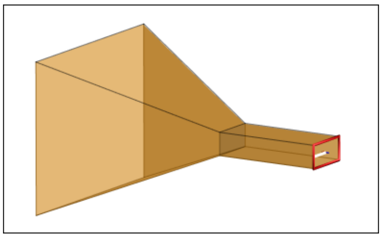


Figure 1-1: Illustration of the horn antenna.

1.2 Before starting the example

Before starting this example, please ensure that the system satisfies the minimum requirements before starting. A user should also ascertain whether the topics presented in this example are relevant to the intended application and FEKO experience level.

The topics demonstrated in this example are:

- General program flow of the FEKO.
- Launching CADFEKO.
- The CADFEKO layout.
- The POSTFEKO layout.
- Viewing the simulated far field and near field results in POSTFEKO.

[1]The demo video is a short movie demonstrating the solution of a horn antenna similar to the example that is shipped with the Altair FEKO installation. It may be found in the `doc` subdirectory of the Altair FEKO installation. For Windows installations, this file is an `.exe` file, which may be executed directly or from the Windows start menu. On Linux installations, the demo is opened inside an `.html` file. This demo video, as well as other FEKO demonstration videos may be viewed on our website.

The requirements for this example are listed below.

- FEKO 2017.2 RELEASE or later should be installed[a] with a valid licence.
- It is recommended that the demo video be watched before attempting this example.
- This example should not take longer than 30 minutes to complete.

[a] See the FEKO Installation Guide to install Altair FEKO.

While working through this example, the steps should be followed sequentially, otherwise explanations may seem to be out of context.

1.3 Different FEKO components and workflow

The components that are most visible to users are CADFEKO, POSTFEKO and the FEKO solver. CADFEKO is the CAD component where the model is created and solution settings are applied. Once the model has been created, it needs to be meshed and then the kernel is run to produce simulation results. The results are then viewed, manipulated and exported in the post processor, named POSTFEKO. After viewing the results it is often required to modify the model again in CADFEKO and then repeat the process until the design is complete. Figure 1-2 illustrates this simple workflow.

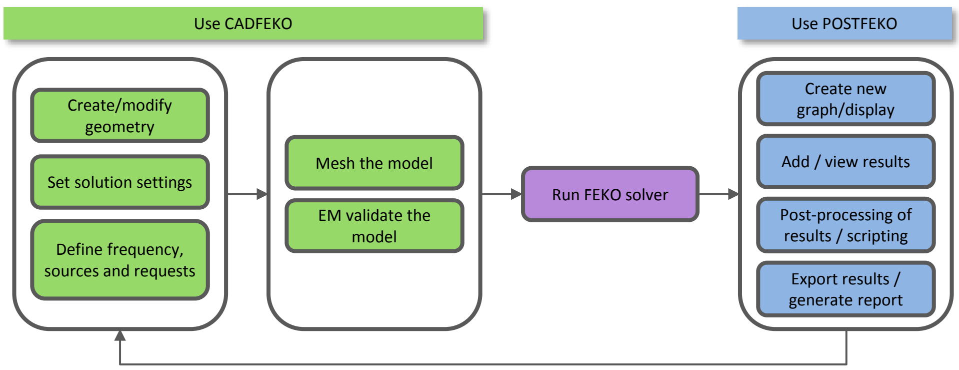


Figure 1-2: Typical FEKO workflow when using CADFEKO, POSTFEKO and the FEKO solver.

A text based model editor, EDITFEKO, is also available for constructing models and solution configurations in a text file. EDITFEKO usage is not covered as part of this example since it should only be required by advanced users who are already familiar with FEKO.

The sections that follow will walk the user through the workflow and introduce the interfaces of the FEKO components as they are encountered. The first component in the workflow is CADFEKO.

1.4 Brief introduction to CADFEKO

When starting a blank instance of CADFEKO or POSTFEKO (no models are loaded), the start page will be displayed, giving quick access to functions that are required most often. The start page allows the user to:

- *Create a new model.*
- *Open an existing model* or select a recently used file from the list.
- Access the various documentation resources. There are links to the PDF documents, demonstration videos as well as a link to our website where other resources are available including demonstration models and articles where FEKO has been used. It is recommended that the videos be watched by first time users.

  The help assistant is available by clicking the green question mark icon at the top right of the start page. The context sensitive help can be launched at any time for context sensitive documentation by pressing <F1> while working in a FEKO component.

All the functionality available on the start page is also available at other locations in the interface, but the start page makes these features readily available when the application is launched.

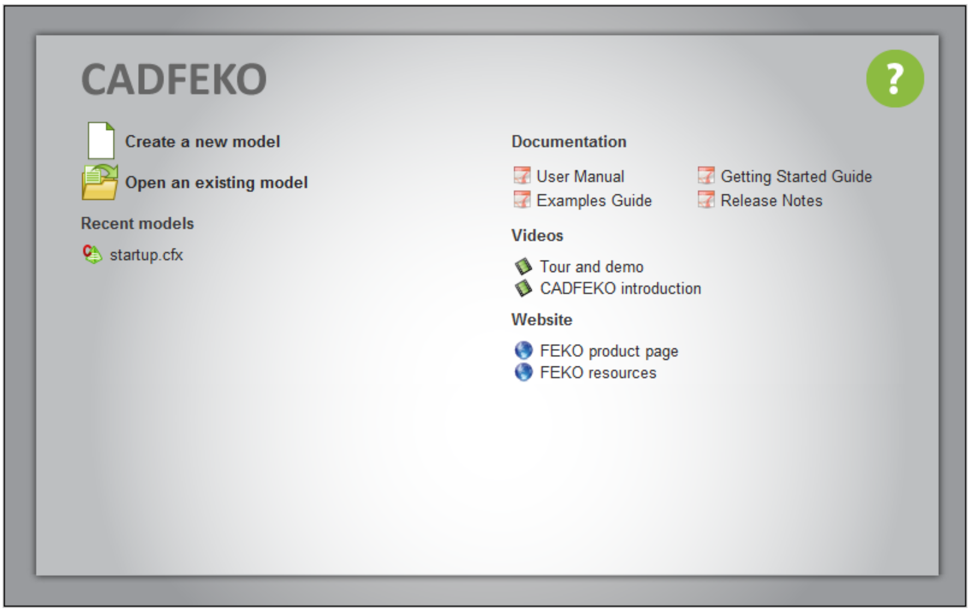
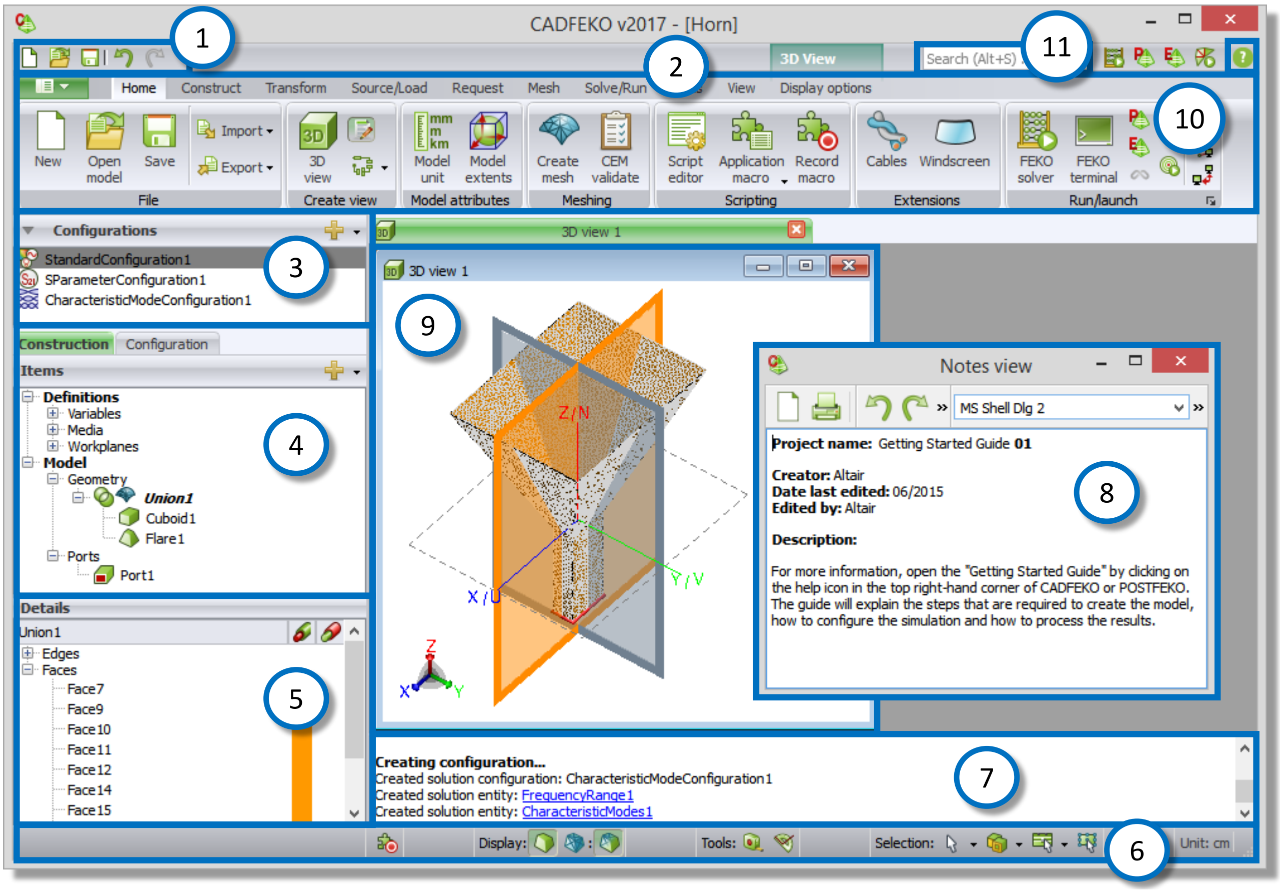


Figure 1-3: The CADFEKO start page.

Application layout

The main elements and terminology used to describe the CADFEKO window will briefly be presented. This terminology will be used extensively in the chapters to follow as well in the manuals. Although most of the components are standard Windows interface components, time will be spent discussing these elements.



1. **Quick access toolbar** These items give the user quick access to controls that are used often. These include the following two lists.

   - Grouped on the left side of the toolbar are model actions that are used frequently.
     - *New model*
     - *Open model*
     - *Save model*
     - *Undo* and *Redo*
   - Grouped on the right side of the toolbar are icons to launch other FEKO components. These icons are located next to the help icon [10] and are collectively referred to as the *Application Launcher*.
     - FEKO solver
     - POSTFEKO (for the display of the results obtained by the FEKO solver)
     - EDITFEKO
     - PREFEKO

2. **Ribbon** The ribbon contains the application menu, default tabs and contextual tabs. A more detailed description of the ribbon and its components is available further on in this section.

3. **Configurations list** The configurations list contains the configurations (standard configuration, multiport S-parameter and characteristic modes) that have been defined in the model. By default a new model starts with a single standard configuration, but simply adding a

multiport S-parameter or characteristic mode will create a new configuration. Multiple configurations allow a user to perform efficient simulations of different configurations (different loads or sources) in a single model.

4. **Model tree** The *Model tree* consists of more than a single tree. The *Construction* tab and *Configuration* tab make two trees available to the user. Definitions such as variables, media and named points are listed in both trees giving easy access to these items in both tabs.

   Construction tree The construction tree is where the user creates and edits the geometry. Ports, mesh refinement rules, infinite planes and arrays can also be constructed.

   Configuration tree The configuration tree displays the selected solution configuration including sources, loads and requests. Configuration settings can be specified globally or specific to a configuration.

5. **Details tree** The details tree contains the geometry object details (wires, edges, faces and regions) for the geometry or mesh part that is selected in the construct tree. Custom solution and mesh settings may be set in the details tree.

6. **Status bar** The status bar gives the user quick access to macro recording, general display settings, tools, selection method and type. It is an "active" status bar since most of the items on the status bar are icons that perform actions or change display settings.

7. **Message window** The message window displays messages regarding user interaction such as geometry creation, meshing, source configuration. It also provides details regarding warnings and error messages. Errors and warnings in the message window will provide links to the corresponding geometry objects in the details tree which resulted in the error or warning.

8. **Notes view** The notes view can be used to document the model details. Additional comments, explanations or descriptions can be added for later reference. The notes view is not displayed by default, but can be activated on the *View* tab.

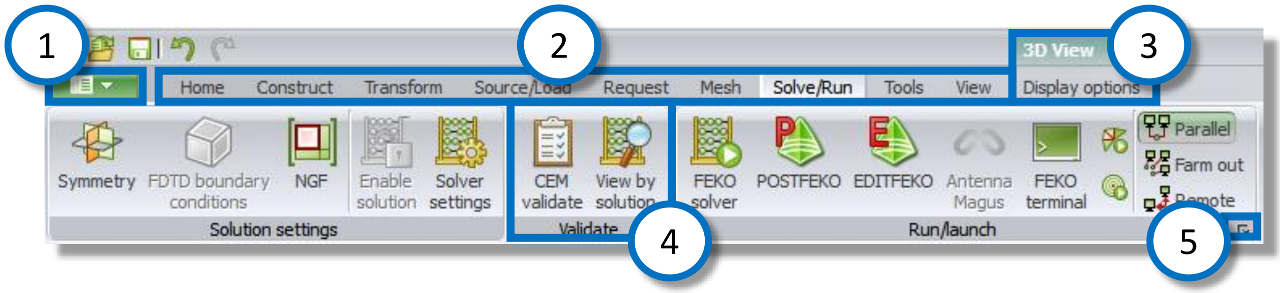
9. **3D view** The 3D view enables the user to visualise the geometry and solution settings (such as far field requests). Note that the configuration items, such as the requests, are displayed for the currently selected configuration and that they are not displayed when the *Construction* tab is selected. This allows the user to concentrate on the CAD when the *Construction* tab is selected. Additional visualisations such as cutplanes and symmetry can also be displayed.

10. **Help** The *Help* icon gives the user quick access to the FEKO manuals. Context sensitive help is available in all FEKO Suite GUI components by pressing <F1> at any time.

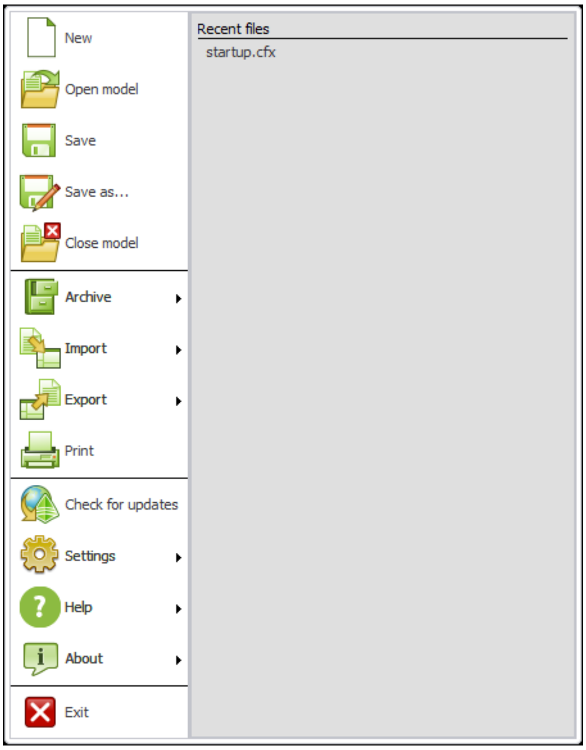
11. **Search bar** The search bar enables the search for a specific action or keyword in CADFEKO. Entering a keyword in the search bar will populate a dropdown list of actions as well as the location of the particular action on the ribbon or context menu. Clicking any one of the items in the list will execute the action.

The ribbon menu

The ribbon consists of several elements. Please take note of the terminology as it will be used extensively in the chapters to follow.



1. **Application menu** The application menu appears when a user clicks the application menu button. The application menu is similar to the standard file menu of an application. It allows saving and loading of models, import and export options as well as giving access to application wide settings. The FEKO updater can be launched from the application menu and it also provides a list of recently accessed files.



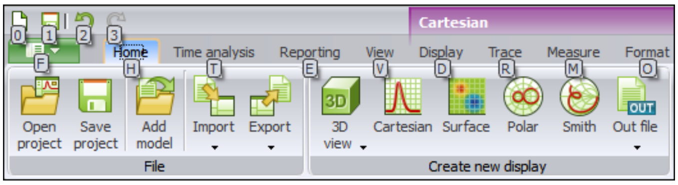
2. **Default tabs** The default tabs are always visible (when enabled) and contain general actions. Most of the tabs in CADFEKO are default tabs.

3. **Contextual tabs** The contextual tabs are context sensitive. These tabs are displayed and changed depending on the currently selected object. The 3D view and schematic view have context sensitive tabs in CADFEKO. A coloured tab marker bar above the tabs indicate the current context.

4. **Ribbon group** Similar actions or commands are contained in a group within a tab.

5. **Dialog launcher** Clicking the dialog launcher will launch a dialog with additional settings that relate to that group. Advanced options that are not used regularly may be found by clicking the dialog launcher button.

The ribbon may also be navigated by means of the keyboard by using the keytips. When pressing and releasing the <Alt> or <F10> key, the tab keytips are displayed. Typing the indicated keytip will open the tab or perform the selected action.



1.5 Opening the first model in CADFEKO

Now that the basic components and structure of CADFEKO have been presented, open a model in CADFEKO. The CAD model is stored in a `*.cfx` file. Start CADFEKO by pressing the Windows *Start* button, typing 'CADFEKO' and running the application from the list of filtered options. The model referred to in this example can be found in the

```
examples\GetStarted_models\Project1-Rectangular_Horn_Antenna
```

directory of the Altair FEKO installation or downloaded from our website.

From the start page in CADFEKO, click *Open an existing model* and select the `Horn.cfx` model. The geometry of the horn should now be visible on the screen. The model can be used to investigate the trees and lists to the left of the CADFEKO interface. The contents of these will be discussed in more detail in the sections that follow.

Contents of the model

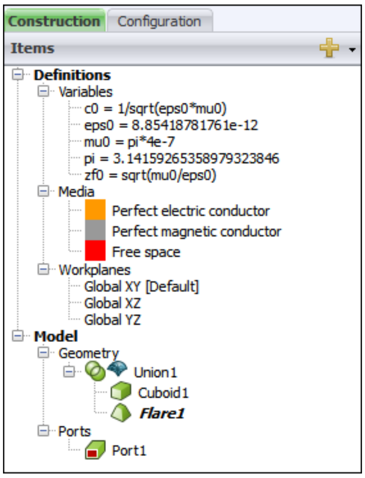
Since the horn example consists of a single configuration, the configuration list will not be discussed in detail. The first list on the left hand side of CADFEKO is the configuration list. For the simple horn model it contains only a single standard configuration.

The model tree consists of the following tabs.

- *Construction* tab with an associated *Details* tree
- *Configuration* tab

Each tab contains trees that are relevant to either the model's construction or its configuration settings. Some entries are displayed on the trees of both tabs (such as variables, media and workplanes).

Construction tab



The *Construction* tab contains the geometry representation of the current model. It contains a list of the definitions such as variables, media and workplanes. The model CAD (geometry or mesh) is also listed in the tree.

Other elements may be defined from its context menus and will be displayed in the tree once an item has been created. These include named points, cable definitions, mesh refinements, plane/ground settings and antenna arrays.

Optimisation searches and associated masks, parameters and goal functions will be displayed in the tree.

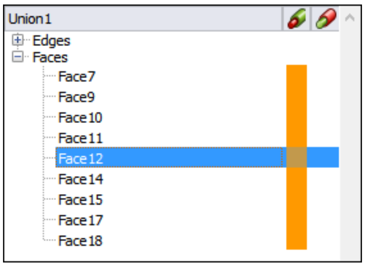
The *Model* branch is a visualisation of the geometry creation hierarchy. Parts are derived from existing ones (for example, the individual parts used in a Boolean operation or the original part before a split operation). The original (parent) parts are removed from the top level of the model and listed as sub-branches under the new part in the model tree.

The term *part* is used for highest-level items. These can be at the root level under *Model* or in the top level of an assembly. The model parts are visible in the 3D view.

Right-clicking any entry in the model tree will open an appropriate context menu. Double-clicking an item in the model tree will display the *Properties* for that item.

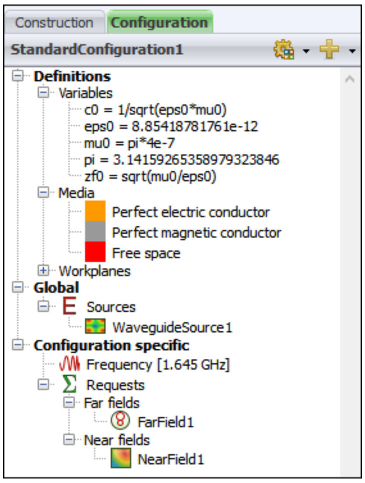
When items are hidden, they are displayed with greyed icons in the model tree, but will not be displayed in the 3D view. Hiding parts simplifies working with complex models containing a large number of parts.

When a geometry or mesh part is selected on the *Construction* tab of the model tree, additional information is displayed in a window positioned just below. This is known as the *Details tree*.



If the *Union1* part is selected, the details tree contains a tree with two branches, namely *Faces* and *Edges*. If the horn geometry contained a closed volume, a *Regions* branch would also be visible and if wires were present, a *Wires* branch would be visible. By selecting wires, edges, faces or regions in this tree, local properties such as mesh size, coatings and solution method can be set.

Configuration tab



The *Configuration* tab consists of the *Global* and *Configuration specific* model settings and requests.

The following *Global* model settings may be defined from its context menus: solver settings, the global frequency, global sources, global loads, global networks and the global power settings.

The following *Configuration specific* settings may be defined from its context menus: requests, frequency per configuration, sources per configuration, loads per configuration and power per configuration.

Setting the solution control

'Ports' are required to add sources and loads to the geometry. Ports define the positions on the model where loads and sources can be attached. In this model, a waveguide port is added to the end of the waveguide section feeding the horn. Ports can be added from the *Ports* group on the *Source/Load* tab. A source can be added to the waveguide port by clicking the *Waveguide source* icon (*Sources on ports* group).

The simulation frequency can also be set on the *Source/Load* tab under the *Settings* group.

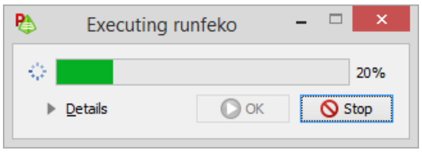
Requests can be added from the *Request* tab. Both far field and near field requests can be made from the *Solution requests* group.

The model contains symmetry that can be used to decrease the required memory and solution time. Symmetry is a model wide setting and can be set on the *Solve/Run* tab under the *Solution settings* group. Two planes of symmetry are present. Magnetic symmetry can be applied in the YZ plane (or the X=0 plane), while electric symmetry can be applied to the XZ plane (or the Y=0 plane). Note that nothing in the model needs to be changed to allow symmetry to be utilised, except that the model must be remeshed.

1.6 Simulating the model

Once the model preparation is complete (geometry, sources, calculation requests and mesh are completed), the solution is obtained by running the FEKO solver.

For this example, the solution is provided to save time. Before the model can be solved, it must be meshed. The solver is invoked by clicking the *FEKO solver* icon on the *Solve/Run* tab.



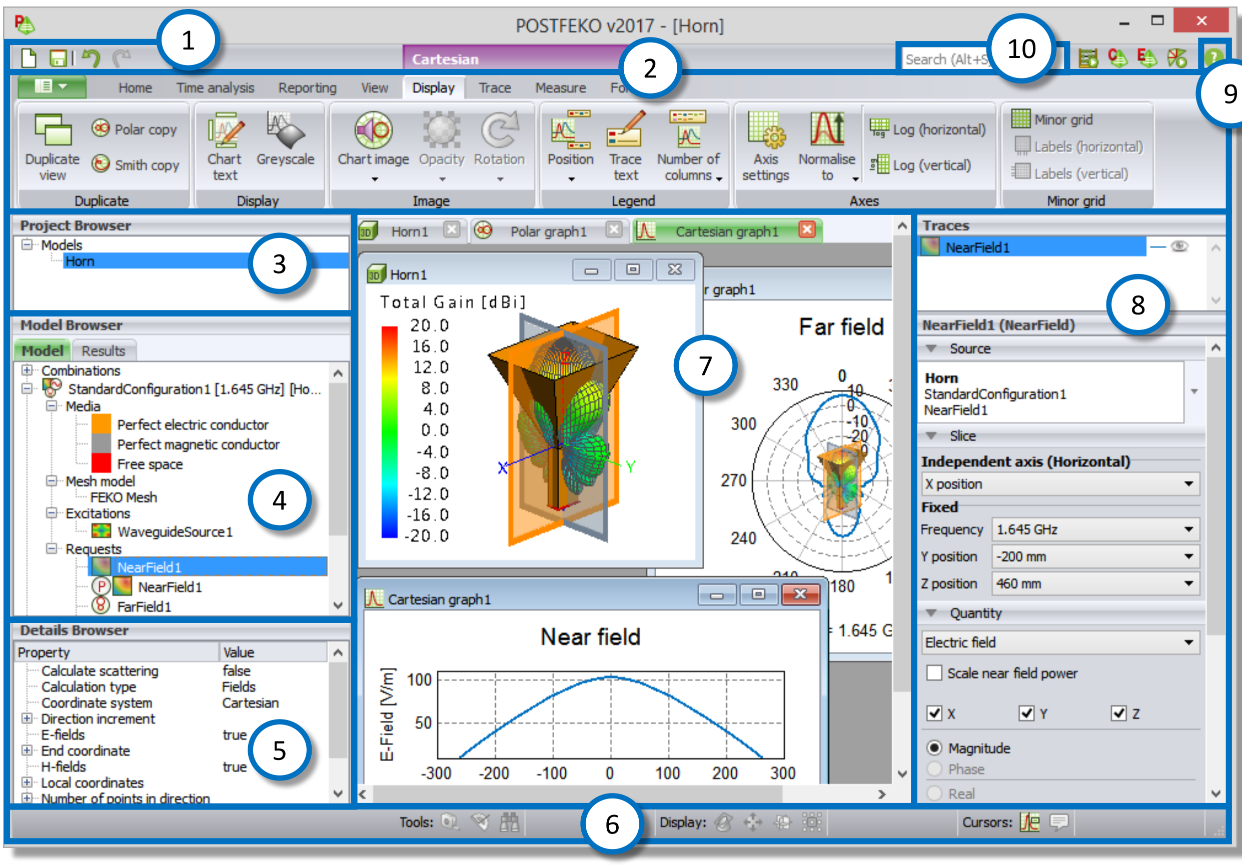
The run dialog will be displayed showing progress information as well as any warnings or errors that are encountered during the simulation. No warnings or errors should be displayed for this model. While the simulation is running, click *Details* to see the full text output produced during the simulation. Also note that there are tabs where notes, warnings and errors will be displayed allowing users to quickly see any problems that may have been encountered.

1.7 POSTFEKO overview

POSTFEKO is used for validating the simulated model to ensure that it represents the intended model and for the visualisation of computed results. POSTFEKO is launched from the *Solve/Run* tab in CADFEKO. Note that the FEKO applications can also be launched from the application launcher in the top right-hand corner of the screen. Launching POSTFEKO from CADFEKO opens the correct model and does not display the start page.

The POSTFEKO window consists of the *Ribbon* at the top and the *Result palette* to the right. The ribbon contains the application menu, default tabs, contextual tabs and the *Quick access* toolbar. The configuration of the window is similar to the CADFEKO main window as described in the previous chapter.

The various main elements and terminology of the POSTFEKO window will be briefly described, but items that are the same in CADFEKO ([1], [2], [6], [9], [10]) are not repeated here.



3. Project browser The project browser lists all the models that are loaded in the current project as well as imported data, stored data and scripted data. The visibility of the project browser can be changed by selecting the *View* tab and clicking the *Project* icon.

4. Model browser The model browser displays information pertaining to the selected model in the Project browser. The information includes optimisation, media, meshes, sources and requests.

5. Details browser The details browser shows in-depth detail of any component selected in the model browser.

7. 3D View/2D graphs The 3D view enables the user to visualise the geometry, mesh, solution settings as well as 3D results. The 2D graphs enable the user to view 2D results on a Cartesian, polar, Cartesian surface graph or Smith chart. The 2D and 3D views each have their own context-sensitive ribbon tabs. The window tabs of the 2D and 3D views may be re-ordered by simply dragging it to the desired location.

8. Result palette The result palette enables a user to apply custom visualisation settings to 3D results or 2D traces by customising the graph contents.

9. Help The *Help* icon gives the user quick access to the FEKO manuals. Context sensitive help is available in all FEKO Suite GUI components by pressing <F1> at any time.

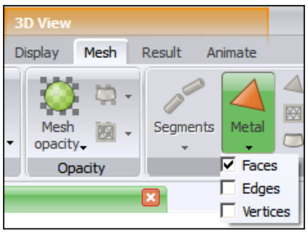
10. Search bar The search bar enables the search for a specific action or keyword in CADFEKO. Entering a keyword in the search bar will populate a dropdown list of actions as well as the location of the particular action on the ribbon or context menu. Clicking any one of the items in the list will execute the action.

1.8 Viewing and validation of the model

POSTFEKO provides visualisation tools and settings to check that the model was created as intended. In this section a number of features in POSTFEKO are highilighted to help the user explore. Users are encouraged to have a look at and experiment with other features that are available while navigating the ribbon tabs.

The horn model should be open showing the model and symmetry planes in the 3D view. The distance measurement tool can be used to validate the specifications (dimensions) of the horn.

Display settings



By default the mesh edges will not be shown. The option to display the triangle mesh edges may be set by clicking the *3D View* context, *Mesh* tab (*Visibility* group) and selecting *Edges* from the *Metal* dropdown menu.

The *Symmetry* display can be disabled by selecting the *Display* tab in the *3D View* context and clicking the *Symmetry* icon (*Method display* group). The toggle icon for symmetry display is also available on the status bar.

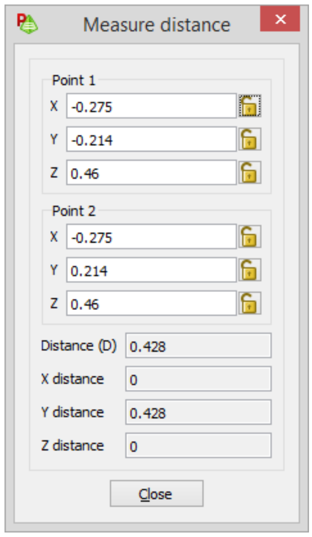
Sources in the model are displayed by default. The display of the source may be disabled by selecting the *Display* tab in the *3D View* context and clicking the *Sources* icon (*Entities* group).

Restore the default zoom by pressing <F5>. The *Zoom to extents* icon can be found on the *View* tab.

Mesh and size validation

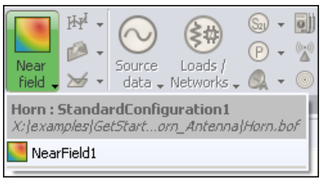
The next step in the validation process is to visually check the dimensions. One option for a quick validation is to enable the tick marks on the axes.

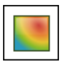
Select the *Display* tab of the *3D View* contextual tab and click the *Tick marks* icon.



For more precise validation of the dimensions, select the *Mesh* tab of the *3D View* contextual tab and click the *Measure distance* icon to launch the *Measure distance* dialog. To measure a distance between two points, hold down <Ctrl><Shift> and left-click on the first and then on the second point. The distance will be displayed at the bottom of the *Measure distance* dialog.

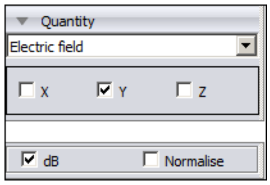
1.9 Near field results (3D)

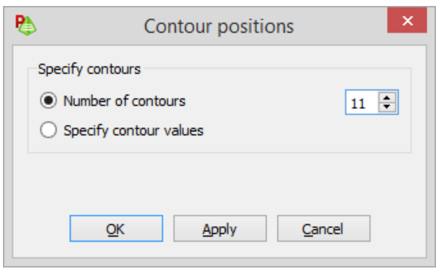




The near field calculation which was requested in this example has been performed and stored. To display this data in the 3D view, select the *Home* tab and click the *Near field* icon. From the dropdown menu select *NearField1*.

For the purposes of this example, the magnitude of the E_y component of the field should be displayed in dB, together with a legend and contour lines for better visualisation. In the *Result palette*, *Quantity* section, unselect the *X* and *Z* checkboxes. Select the *dB* checkbox.

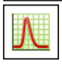




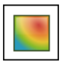
To place the contour lines on the display, select the *Result* tab. Click the *Positions* icon (*Contours* group) and select *Number of contours*. Set the number of contours to 11 and click the *OK* button. To enable the display of the contours, click the *Show contours* icon (*Contours* group).

1.10 Near field results (2D)

The near field results can be displayed on a 2D graph for comparison purposes.

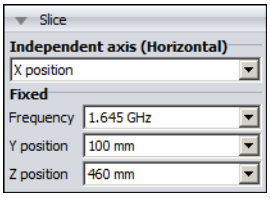


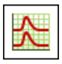
Select the *Home* tab and click the *Cartesian* icon to create a 2D graph. (Near field data can also be displayed on a Polar graph if the near field points were requested in spherical or cylindrical coordinates.)



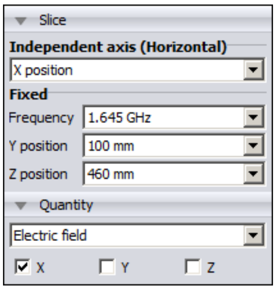
Click the *Near field* icon (*Add results* group) and from the dropdown list, select *NearField1*. This example compares the E_y and E_x components along the X direction (orthogonal to the polarisation) about a quarter wavelength from the edge of the horn.

In the *Result palette*, *Slice* section, set the *Independent axis* to *X position*. In the *Fixed* section, set the *Y position* to 100 mm and *Z position* to 460 mm. Unselect the *X* and *Z* checkboxes, so that only the *Y* component is selected.



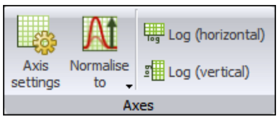


To add the second trace to the graph, select the *Trace* tab of the *Cartesian* contextual tab. Click the *Duplicate* icon. (We could also have added the second trace from the ribbon in the same way as the first trace.)

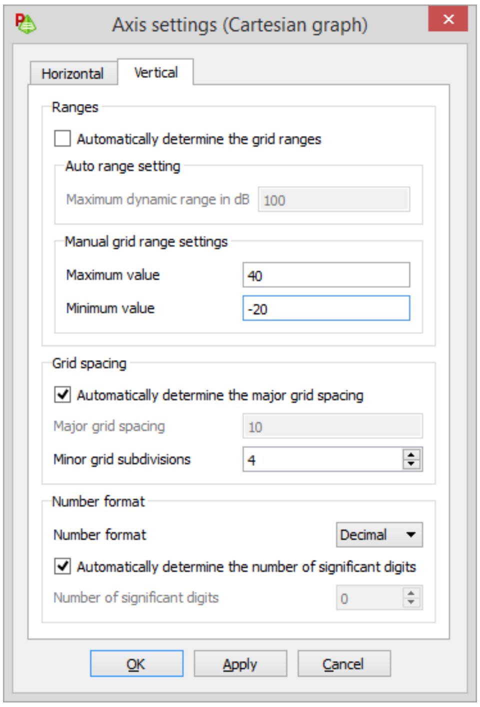


With the trace *NearField1_1* selected in the *Result palette*, unselect the *Y* and select the *X* component in the *Quantity* section. This will result in only the E_x component of the field being displayed.

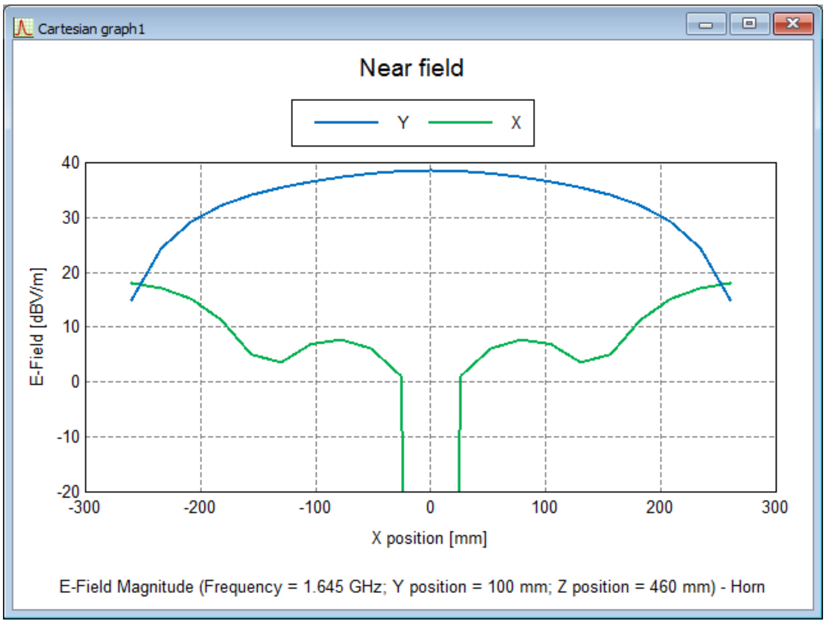
Over most of the aperture E_x is much smaller than E_y. It is therefore recommended to set the vertical axis to dB. Select both traces and check the dB checkbox in the *Quantity* section.



The minimum and maximum values for the vertical axis may be changed by selecting the *Display* tab of the *Cartesian* contextual tab. Click the *Axis settings* icon (*Axes* group) which will launch the *Axis settings (Cartesian graph)* dialog.



On the *Axis settings (Cartesian graph)* dialog select the *Vertical* tab. Unselect the *Automatically determine the grid ranges* checkbox. Set the *Minimum value* of the vertical axis to −20 and the *Maximum value* to 40.

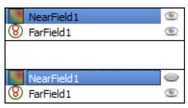


Settings related to the format of the font, colour and line style, may be modified on the *Format* tab.

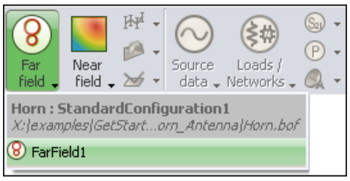
1.11 Far field results

The far fields on a full sphere were requested in the solution control phase of this example. This section explains the procedure to plot these results in a 3D view and on a polar plot.

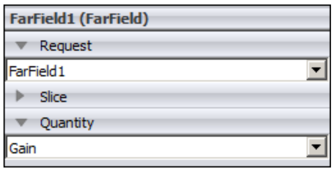
In order to display the results in the 3D view, the 3D view must be the active window. Select the 3D view window. The near field result currently displayed in the 3D view can be hidden by clicking the eye icon next to the trace *NearField1* in the *Result palette*.



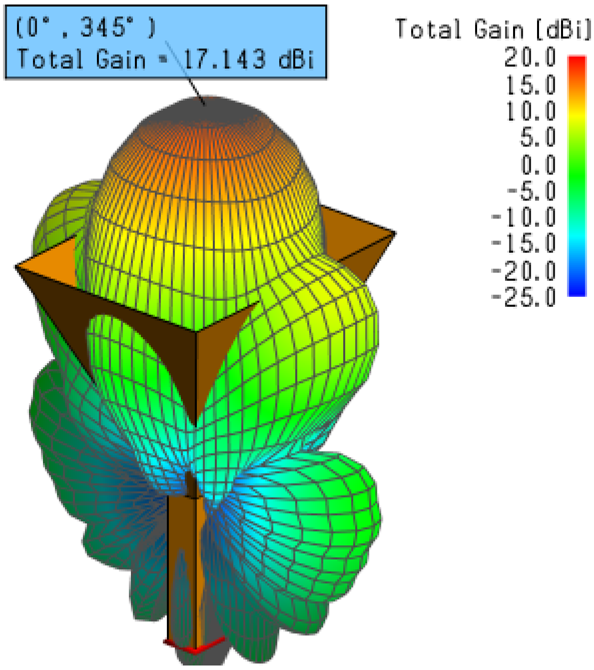
When a result is hidden, a closed eye icon next to the result will indicate that it is hidden.



Select the *Home* tab and click the *Far field* icon (*Add results* group). From the dropdown list select *FarField1*. The annotation can be added by holding down <Ctrl><Shift> and left-clicking when the annotation is displayed in the desired location.

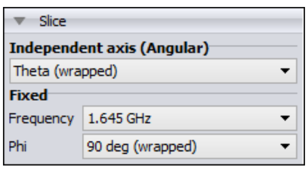


Note that the fields are immediately displayed in the 3D view. In the *Result palette* set the *Quantity* to *Gain*. Also check the *dB* checkbox to set the scale to dB.



The size of the far field compared to the geometry size can be changed by selecting the *Result* tab of the *3D View* contextual tab. Click the *Size* icon (*Rendering* group) and select the *Custom...* option from the dropdown list. Specify the size as 70%.

Far field radiation patterns are often displayed on 2D polar graphs. Since full 3D data is requested in this simulation, 2D cuts can be extracted. To create a new polar graph, click the *Polar* icon (*Home* tab, *Create new display* group). Click the *Far field* icon (*Home* tab, *Add results* group) and from the dropdown list select *FarField1*. In this case it is requested that the far field gain be plotted in the YZ plane on a polar graph. This corresponds to plotting the data with respect to θ at a constant value of $\phi = 90°$.



In the *Result palette*, *Independent axis* section, select *Theta (wrapped)*. Select the value for *Phi* as *90 deg (wrapped)*.

Due to the sampling chosen when the calculation was requested, the wrap option must be selected to complete the pattern. Set the *Quantity* to *Gain* and select the *dB* checkbox.

1.12 Closing remarks

This introductory example has shown aspects of the user interface of FEKO, from *pre*-processing through to *post*-processing, using an existing model of a horn antenna supplied with the Altair FEKO installation. The next example takes the user through the process of creating the geometry and also shows how to model dielectric regions.

2 Getting started: Creating models in CADFEKO

2.1 Example overview

This example uses a completed model to familiarise the user with model creation in CADFEKO. It is intended for users with no or little experience with CADFEKO. In this example no electromagnetic solution is performed and no results are presented – this example illustrates the creation of models from geometry parts, the combining of primitives and transforming a primitive to a different location in the model. This example does not use the fastest, most effective way to create the particular geometry, but instead illustrates the tools available in CADFEKO that allows users to create complicated geometrical structures.

Figure 2-1 shows an illustration of the geometry that is created in this example.

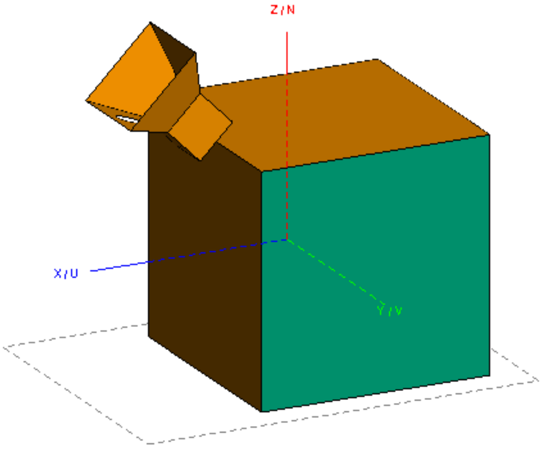


Figure 2-1: Illustration of the geometry created in this example.

2.2 Before starting the example

Before starting this example, please ensure that the system satisfies the minimum requirements before starting. A user should also ascertain whether the topics presented in this example are relevant to the intended application and FEKO experience level.

The topics demonstrated in this example are:

- Launching CADFEKO
- Creating geometry from primitives
- Combining geometry to create more complicated objects (union, sweep, subtract)
- Using workplanes, snapping and moving objects in and out of operators
- Finding and fixing suspect items
- Setting material properties
- Relocating objects

The requirements for this example are:

- FEKO 2017.2 RELEASE should be installed[a] with a valid licence.
- It is recommended that the demo video be watched before attempting this example.
- This example should not take longer than 40 minutes to complete.

[a]See the FEKO Installation Guide to install Altair FEKO.

While working through this example, the steps should be followed sequentially, otherwise explanations may seem to be out of context.

The model referred to in this example can be found in the

```
examples/GetStarted_models/Project2-Models_in_CADFEKO/Model_creation.cfx
```

directory of the Altair FEKO installation or downloaded from our website.

2.3 Starting CADFEKO

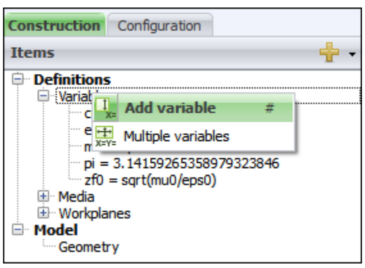
The first step in every FEKO solution is the construction of the model geometry. The geometry is created using the CAD component, CADFEKO. The CAD model is stored in a `*.cfx` file. Start CADFEKO by pressing the Windows Start button, typing 'CADFEKO' and running the application from the list of filtered options. Click the *Create a new model* link on the start page.

2.4 Building a horn

First the horn will be created. It consists of a cube and a flare that have been unioned and the excess faces removed. The cube is not constructed using the cuboid primitive, but rather created using a rectangle and the path sweep operator.

Add variables

Any variables required to build the model can now be defined. Parametric models can be created by using variables. It is the recommended construction method, but it is not compulsory.



In the model tree is a list of predefined variables. Three mechanisms are available to add variables to the model:

On the ribbon, from the *Variable* icon on the *Construct* tab (*Define* group).

On the model tree, from the *Add* menu at the top of the *Construction* tab and *Configuration* tab.

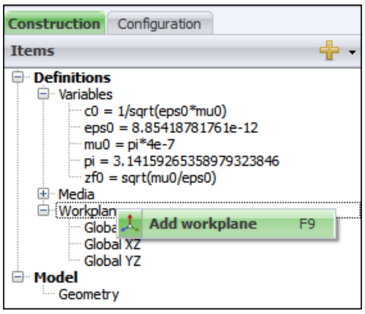
A context menu (right-click menu) is also available on the *Variables* group in the tree.

Variables are created by entering a variable name and expression and clicking the *Add* button (to keep the dialog open for adding more variables) or *Create* (to add the variable and close the dialog). A short comment for the variable can be added to the *Comment* field.

Add the following variables (and optionally the comments):

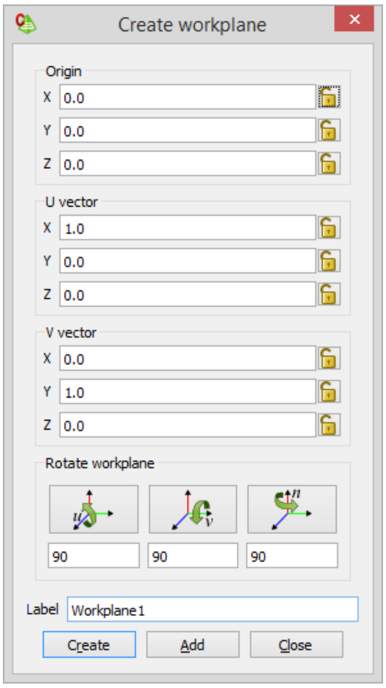
Name	*Expression*	*Comment*
Width	1	Width of rectangle
Length	1	Length of rectangle
BottomDepth	1	Bottom depth of flare
BottomWidth	1	Bottom width of flare
FlareLength	1	Length of flare
TopWidth	2	Top width of flare
TopDepth	2	Top depth of flare

Add a workplane



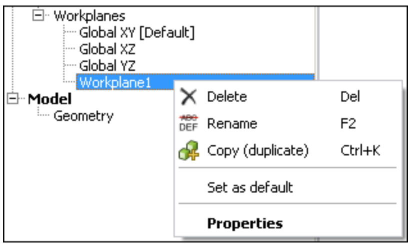
We will now create a custom workplane and set it to the default workplane.

On the *Construction* tab, right-click *Workplanes* and select *Add workplane* from the context menu. As indicated in the context menu, it is also possible to use the <F9> keyboard shortcut to create a new workplane.



Below the *Rotate workplane* groupbox, click once the middle icon indicating a rotation about the V axis. Note how the entries for the *U, V* and *N* vector are modified accordingly. Click the *Create* button.

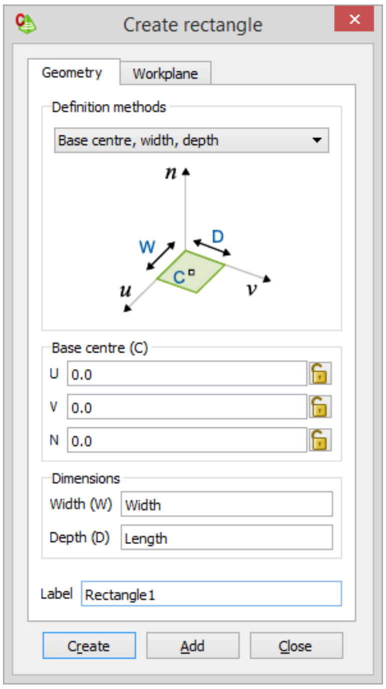
Workplanes are not required that often during construction, but in certain cases they can save time.



The default workplane is used when creating new geometry primitives. For this example we want the new workplane to be the default workplane.

Right-click *Workplane1* to open the workplane context menu. Select *Set as default* from the context menu. The default workplane will be indicated by the text *[Default]* in the model tree.

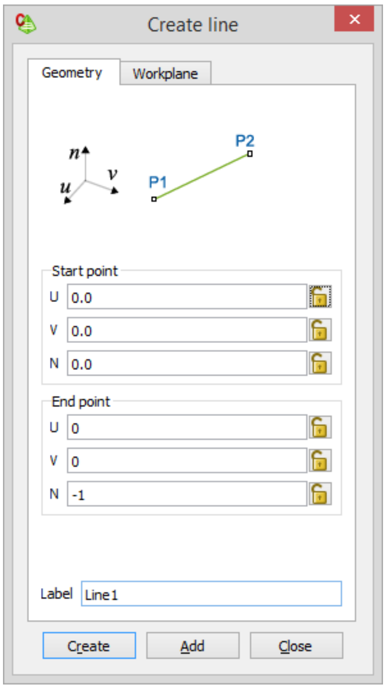
Create rectangle primitive



The next step will be to create a rectangle. Ensure that the *Construct* tab is selected and click the *Rectangle* icon (*Create surface* group) and set the definition method to *Base centre, width, depth*.

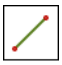
Set the *Width* equal to `Width` and the *Depth* equal to `Length`. Click the *Width (W)* field to make it active. An active field allowing point-entry, is indicated by a yellow background. These variables may either be entered by pressing <Ctrl><Shift> and left-clicking the variable in the model tree (point entry) or by using keyboard input.

Note that default values for primitives are given to enable users to view a preview of the primitive. These values are to be set by the user to the desired value. Click the *Create* button to create the rectangle.



Create line primitive

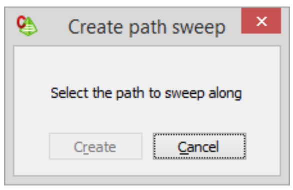
We need to create a line so that we can sweep the rectangle along the line to create the cuboid.



Create a line by clicking the *Line* icon (*Create curve* group). Set the *Start point* to (0,0,0) and the *end point* to (0,0,-1). Click the *Create* button.

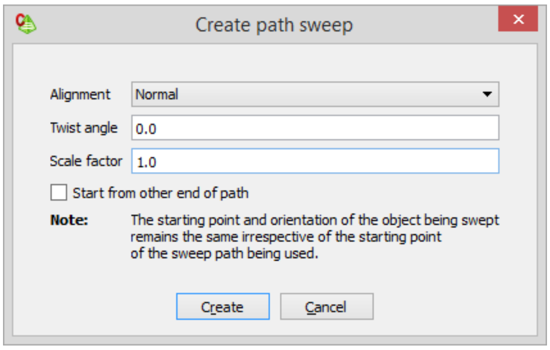
Note that the previously defined workplane is used since it has been set as the default workplane.

Sweep the rectangle along the line



A cuboid will now be created by sweeping the rectangle along a path (the line that was created). The *Cuboid* primitive would usually be used, but for demonstrative purposes, the *sweep* operation is utilised. Select *Rectangle1* in the tree, note that the *Path sweep* icon in the *Extend* group is now enabled. Click the *Path sweep* icon.

The *Create path sweep* dialog will be displayed, requesting that the path to sweep along is to be selected. Click *Line1* in the tree.



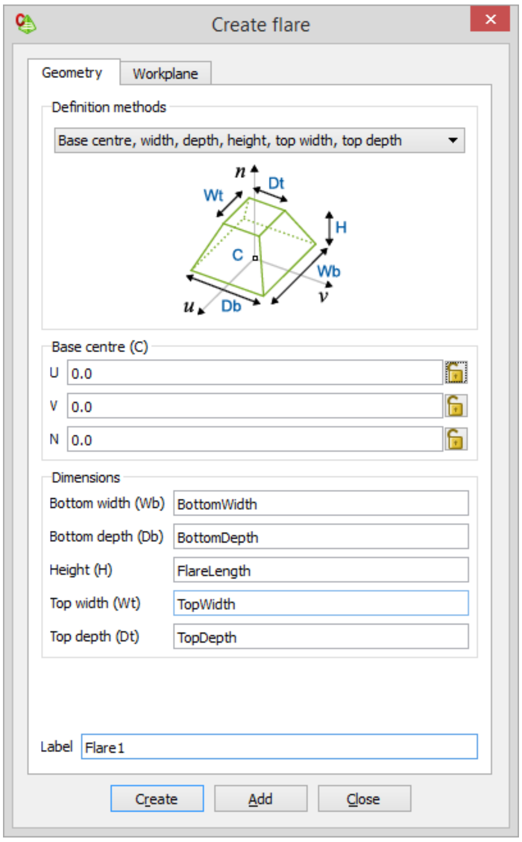
A dialog will be displayed asking the following information from the user: *Alignment*, *Twist angle* and *Scale factor*. Use all the default values for the example, but the user is encouraged to modify the settings and to investigate its effect on the preview in the 3D view. Click the *Create* button.

Create flare primitive

Open the *Create flare* dialog by clicking the *Flare* icon (*Create solid* group). Set the *Definition method* to *Base centre, width, depth, height, top width, top depth*. The use of defined variables are optional. Alternatively, the user may enter values instead of defined values, although parametric models are the preferred methodology for CAD models. These values or the defined variables may be modified at any time. The model will then automatically be updated.

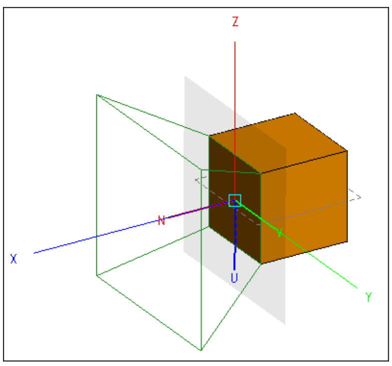
Set the flare dimensions as follows:

Dimension	*Value*
Base centre	(0,0,0)
Bottom width	`BottomWidth`
Bottom depth	`BottomDepth`
Height	`FlareLength`
Top width	`TopWidth`
Top depth	`TopDepth`



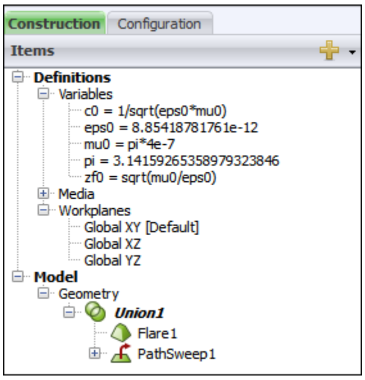
These defined variables may also be entered into their respective fields by pressing <Ctrl><Shift> and left-clicking the defined variable in the tree, under *Variables*.

The model should look as shown in the figure, just before the flare is created by pressing the *Create* button. Click the *Create* button.



Union the flare and the cuboid

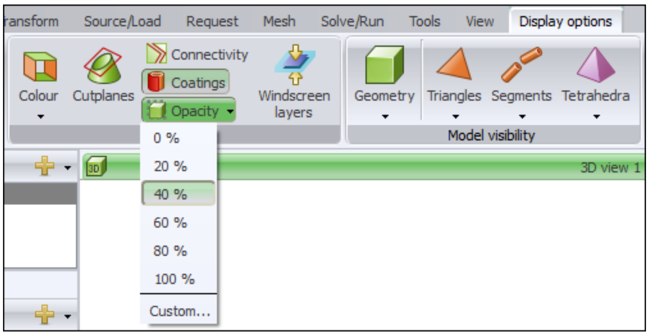
The created flare and swept rectangle now need to be unioned. Note that in CADFEKO the *Union* operation is used to define connectivity between parts. Parts that touch, but are not unioned, are not considered to be physically connected and will not produce the correct mesh.



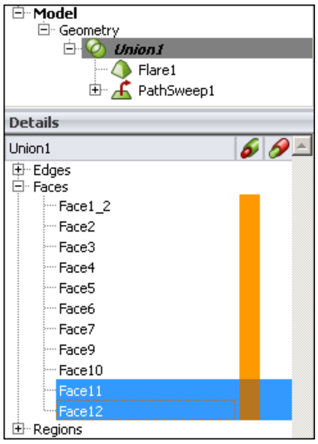
Select the flare and the cuboid (named *Pathsweep1*) in the tree and click the *Union* icon (*Modify* group). Note that it is possible to do a multi select of objects in the model tree (using the <Ctrl> and/or <Shift> keys while selecting).

Tip: The keyboard shortcut for the *Union* operator is <U> and is worth memorising since it is used quite often.

Remove redundant geometry faces

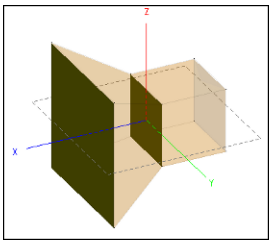


The next step will to be delete the faces set between *Flare1* and *Pathsweep1*. The visualisation of the respective faces will be improved by setting the opacity of the geometry. Select the *Display options* (*3D View* contextual tab) and click the *Opacity* icon (*Style* group) and select for example *40%* from the dropdown menu.



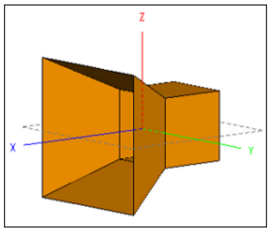
Select *Union1* in the tree. In the details tree, under *Faces*, find the face between the flare and the cuboid as shown in the figure. Right-click the respective face and select *Delete* from the context menu. Also delete the outer face of the flare.

Note that deleting the faces automatically changes the region properties from PEC (solid) to free space (shell).



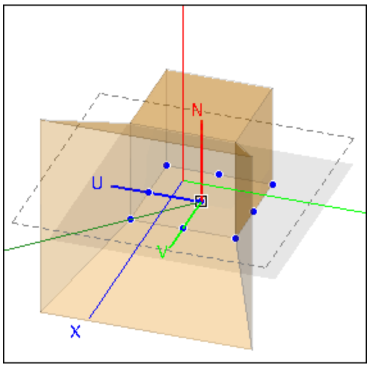
Displayed in the image are the faces that are to be deleted.

Set the *Opacity* back to 100% again. The horn should now look as displayed.

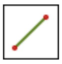


2.5 Add a feed pin to the horn

The next part is to add a wire feed for the model. We will not add the port or the source, but this section does illustrate how a wire (or any other geometry) can be added to an existing union. Set the *Opacity* back to 40% again to allow improved visualisation for the next step.

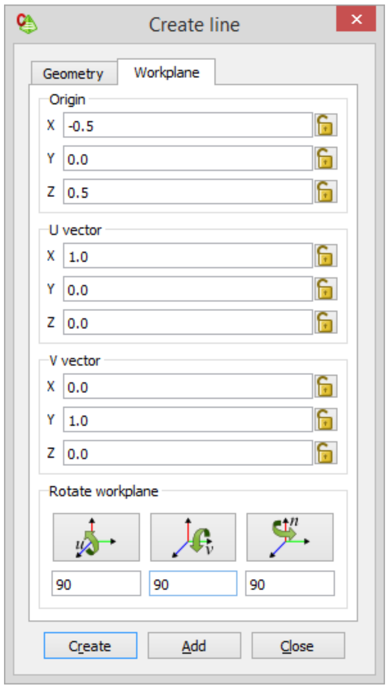


Create the wire



Create a line by clicking the *Line* icon. Click the *Workplane* tab. Hold down <Ctrl><Shift> whilst moving the mouse cursor over the 3D model. Notice that special snapping points to which the new workplane can be snapped to will be indicated by blue dots. Although only special snapping points are indicated by blue dots, it is possible to snap to any point in the 3D view.

Snap the workplane to the centre of the bottom face of the cuboid.



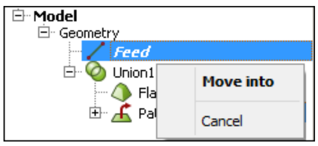
By using the icons in the *Rotate workplane* group, rotate the line's workplane until the values illustrated in the image are entered into the U, V and N vector fields (rotate 90 degrees around the N axis for this specific workplane orientation).

Click the *Geometry* tab. Set the *Start point* as (-0.25,0,0) and the *End point* as (-0.25,0,0.25). This will be the feed element. Modify the *Label* field to Feed. Click the *Create* button.

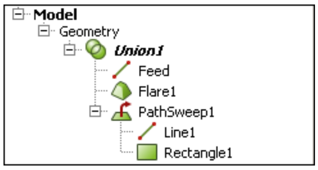
Set the *Opacity* back to 100% again

Union the feed wire with the horn

Select the Feed in the tree and drag it onto *Union1* (also in the tree). A context menu will be displayed, select *Move into*.

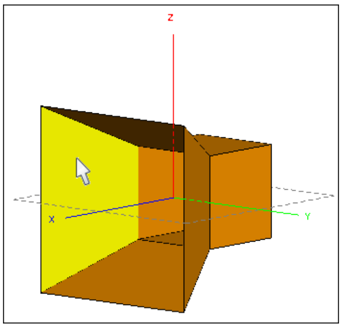


After the *Feed* was moved into the *Union*, the tree should now look as shown in the figure.

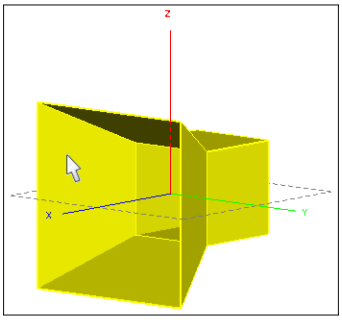


2.6 Selection in the 3D view

The following steps are not required to build the model, but it illustrates how selection in CADFEKO works.

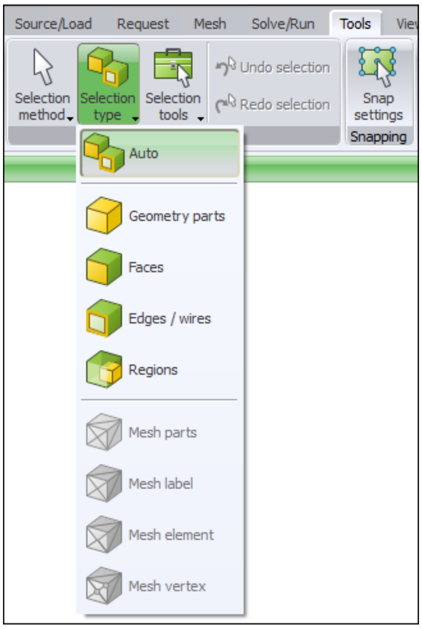


Move the mouse cursor to one of the inside faces of the flare. Left-click the face and notice that the face is highlighted in yellow.



When left-clicking the face again, note that the entire model is highlighted in yellow. The default CADFEKO selection method (*Auto*) cycles through the applicable selection type when repeatedly left-clicking the model. The first left-click highlighted the face of the flare. The second left-click highlighted the flare part.

In CADFEKO, the selection type may be modified by selecting the *Tools* tab and clicking the *Selection type* (*Selection* group). The *Auto* selection type is the default selection type in CADFEKO. Continuously left-clicking a part of the model will cycle through the applicable selection types and highlight the element.



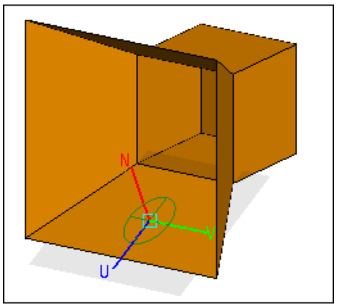
A second method for modifying the selection type is by means of the status bar. Click the *Selection* toolbar and select the required selection type from the menu.

2.7 Cut a hole in a face

Holes in faces and regions are created by first creating the geometry that should be removed and then subtracting the one part from another part.

Create and place the ellipse

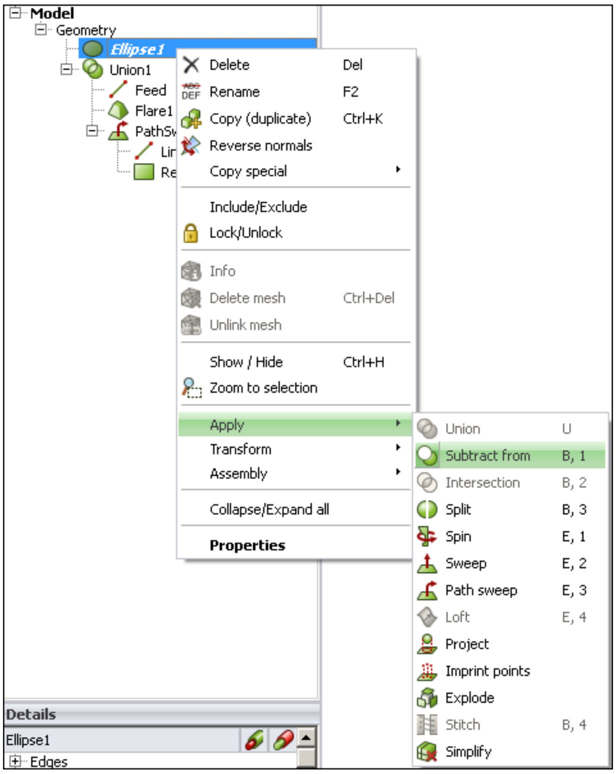
Create an ellipse by clicking the *Ellipse* icon (*Create surface* group). Select the *workplane* tab. The workplane will now be placed on the underside of the flare by snapping it to the surface of the flare.



The fields on the workplane tab should have a yellow background (click one of the *origin* fields if this is not the case). Press <Ctrl><Shift> and move the mouse cursor over the flare. Note the change in the *Origin*, *U* and *V* vector fields as the mouse cursor is moved. Special snap points will be indicated by blue dots.

While holding down <Ctrl><Shift> on the keyboard, move the mouse around until the local workplane is orientated as displayed in the image.

Move the mouse cursor close to an edge, then to the face centre. Now move to a different edge and return back to the face centre. Note how the history of where the mouse cursor was moved to the face centre, affects the orientation of the workplane.



Click the *Geometry* tab and set the dimensions as follows:

Dimension	*Value*
Radius (U)	0.3
Radius (V)	0.2

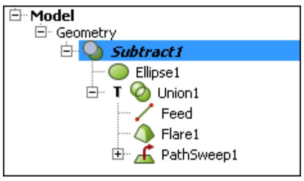
Click the *Create* button.

Subtract the ellipse from the horn

Select *Ellipse1* in the tree and right-click. From the context menu, select *Apply* → *Subtract from* as indicated in the image. It is also possible to launch the subtract operator from the ribbon (*Construct* tab).

A dialog, asking for the object to subtract from will be displayed. Select *Union1* in the tree.

The object from which was subtracted (the target), will be indicated by a T next to *Union1* in the tree (shown in figure).

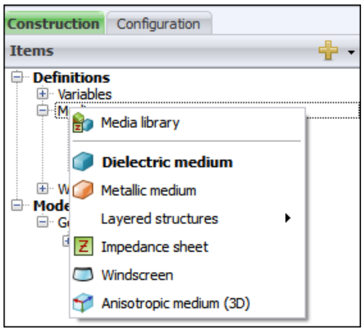


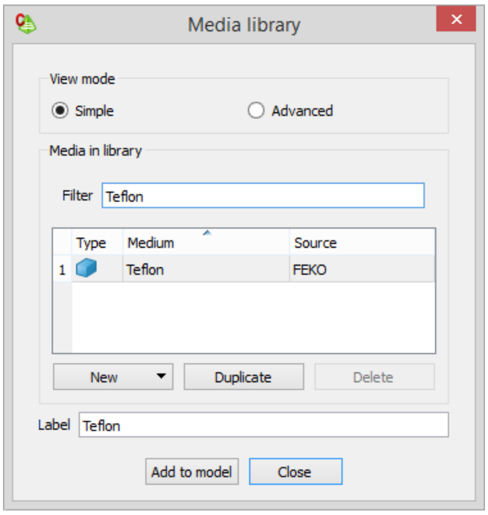
2.8 Create a dielectric object with metal faces

Creating dielectric objects in CADFEKO is simple. The following section will illustrate how a dielectric object is created. We will also set a subset of the faces bounding the dielectric object as metal.

Add a dielectric

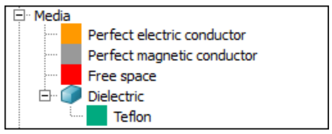
Add a new dielectric medium from the *Media Library* to the model by selecting the *Construct* tab. Click the *Media* menu button (*Define* group) and select *Media Library* from the dropdown menu. Alternatively one can also right-click the *Media* entry in the tree and select *Media Library* from the dropdown menu.





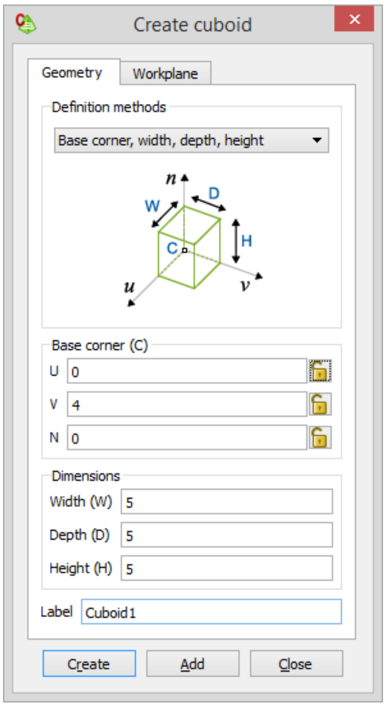
The dielectric object will be defined as consisting of the dielectric material *Teflon*. In the *Filter* editbox, type *Teflon*. At the bottom of the dialog, click *Add to model* to add the dielectric from the library to the CADFEKO model.

After the dielectric is added to the CADFEKO model from the media library, it will be displayed in the tree under *Dielectric* group. The media library can now be closed.



Reset the global workplane

Next, set the *Global XY* workplane as the default workplane. Right-click the *Global XY* and select *Set as default* from the context menu.



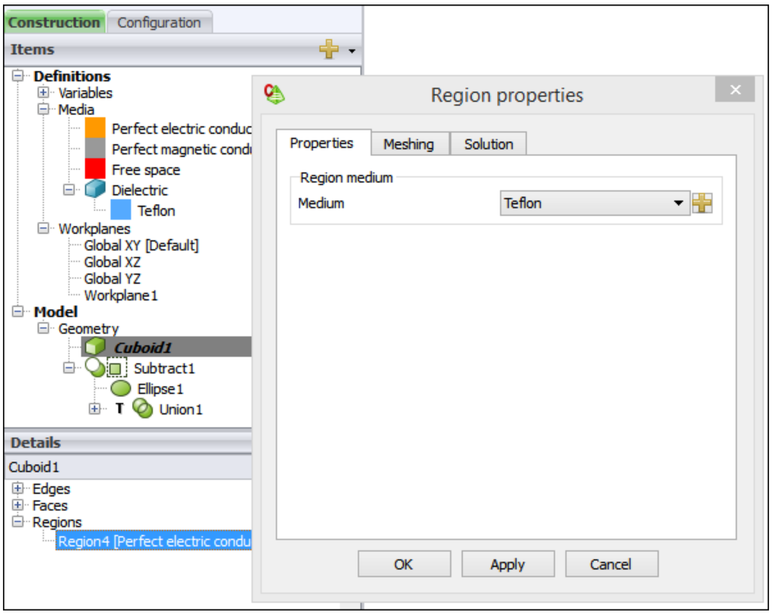
Create a dielectric cuboid

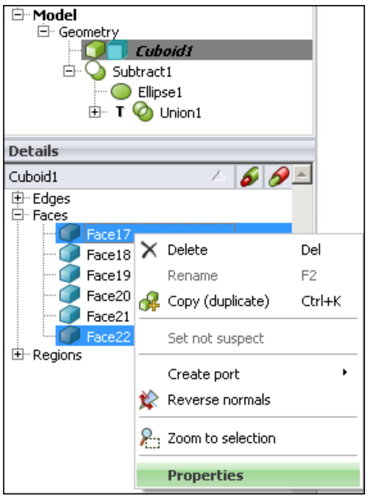
Create a cuboid by clicking the *Cuboid* icon.

Select the *Base corner, width, depth, height* definition method and set the following dimensions:

Dimension	*Value*
Base corner	(0,4,0)
Width	5
Depth	5
Height	5

Select the *Cuboid1* part in the tree. Now right-click the region in the details tree and open the *Properties* dialog. Set the *Region medium* to *Teflon*.

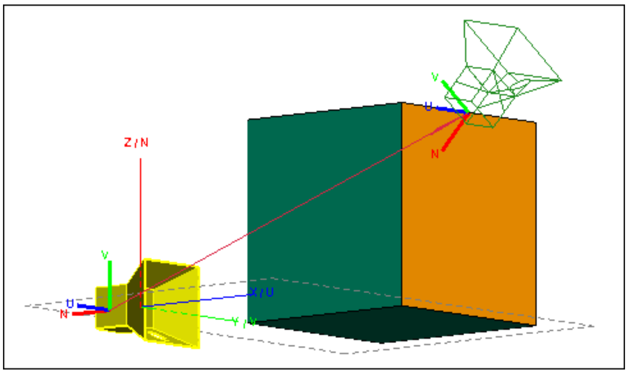




Change faces to metal

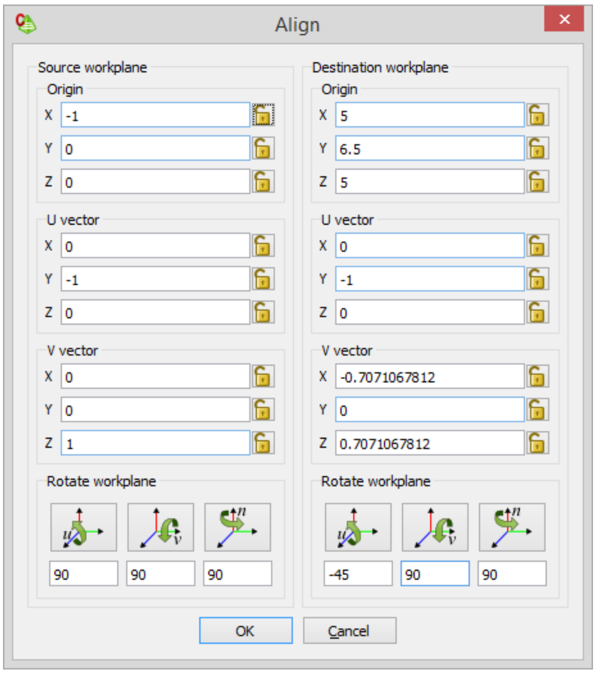
Select *Cuboid1* in the tree and select its corresponding front face (the face at X/U=5) and upper face (the face at Z/N=5) in the details tree. Right-click and select *Properties* from the context menu. At the *Face medium*, set the *Face type* to *Perfect electric conductor*.

2.9 Position the horn on another object



The face centre of the horn will now be aligned on the front upper edge of the cuboid.

Click *Subtract1* in the model tree and select the *Transform* tab. Click the *Align* icon (*Transform* group) to display the *Align* dialog.



Place the *Source workplane* by holding down the <Ctrl><Shift> keys to snap to the centre of the back face of the horn. Click one of the *Origin* fields in the *Destination workplane* group to activate (indicated by yellow background) the destination workplane. The destination workplane is set on the top and nearest edge of the cuboid.

The cuboid should be placed at an angle of 45 degrees to the cuboid. Rotation of the flare may be accomplished by rotating around the U axis and/or V axis and/or the N axis, depending on the direction of the original placement of the source workplane. For the rotation of 45 degrees, enter the 45 in the respective axis and press the rotation icon.

Click the *OK* button to apply the transform and close the dialog.

Union *Cuboid1* and *Subtract1* by selecting them both in the tree and clicking the *Union* icon.

2.10 Closing remarks

This introductory example has shown aspects of model creation in CADFEKO. The next example takes the user through the process of using multilayer planar substrates, using symmetry planes to reduce the required resources, using adaptive frequency sampling to obtain continuous data and viewing the input impedance in POSTFEKO.

Note again that this was not a practical example that can be simulated, but rather an example that illustrates the power of CADFEKO when creating complex models. The other examples are practical examples that can be simulated.

3 Getting started: A patch antenna

3.1 Example overview

A patch antenna designed to operate close to 2.8 GHz will be modelled in this example. The model is first constructed as a patch on an infinitely large substrate since it is quick to create and to simulate. The antenna is then adapted so that it has a finite substrate, making it a more realistic model. Figure 3-1 is an illustration of the patch with a finite ground that we are going to construct.

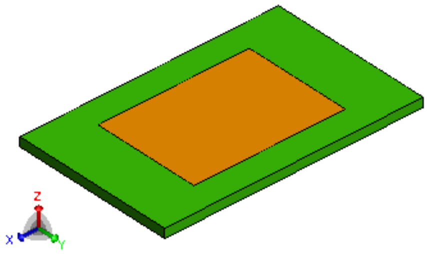


Figure 3-1: Illustration of a patch on a finite substrate.

3.2 Before starting the example

Before starting this example, please ensure that the system satisfies the minimum requirements before starting. A user should also ascertain whether the topics presented in this example are relevant to the intended application and FEKO experience level.

The topics demonstrated in this example are:

- Use of a multilayer planar substrate.
- Modelling of a finite sized dielectric substrate.
- Comparison of simulation time and resource requirements using different modelling techniques.
- Usage of symmetry planes to reduce the required resources.
- Adding a voltage source to a wire segment.
- Using adaptive frequency sampling to obtain continuous data.
- Viewing the simulated input impedance in POSTFEKO.

The requirements for this example are listed below.

- FEKO 2017.2 RELEASE or later should be installed[a] with a valid licence.
- It is recommended that the demo video be watched before attempting this example.
- This example should not take longer than 60 minutes to complete.

[a]See the FEKO Installation Guide to install Altair FEKO.

While working through this example, the steps should be followed sequentially, otherwise explanations may seem to be out of context.

The models referred to in this example can be found in the

```
examples/GetStarted_models/Project3-Patch_Antennas
```

directory of the Altair FEKO installation or downloaded from our website.

3.3 Patch on infinite substrate

A small patch antenna on an infinitely large multilayer substrate is constructed in CADFEKO before running the FEKO solver. The results from the FEKO solver are then viewed in POSTFEKO.

The first step in every FEKO solution is to construct the model. Start by launching CADFEKO which then opens with the start page. Click *Create a new model* to create a new model. The model is created using CADFEKO and stored in the `*.cfx` file.

3.3.1 Creating the model

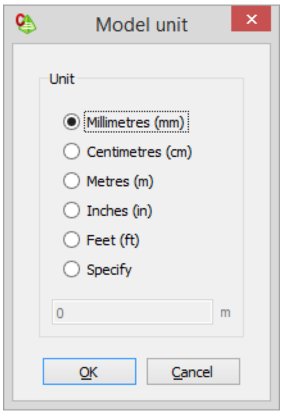
The model creation steps that we are going to perform can be summarised as:

- Set the model unit to millimetres.
- Add variables that define the model geometry and material parameters.
- Add a new dielectric medium type to the model.
- Create the patch.
- Create the planar multilayer substrate.
- Create the feed pin to excite the patch.
- Union the geometry.
- Add a port and voltage source on the feed pin.
- Set the solution frequency.
- Set the symmetry plane.

- Mesh the model and run the FEKO solver.

CADFEKO should now be open with an empty model. Let's start construction of the model.

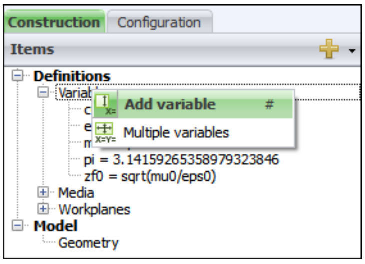
Set the model unit



The default unit length in CADFEKO is metres. Since the patch that we are going to build is small, we should set the model unit to millimetres. All dimensions can then be entered in millimetres.

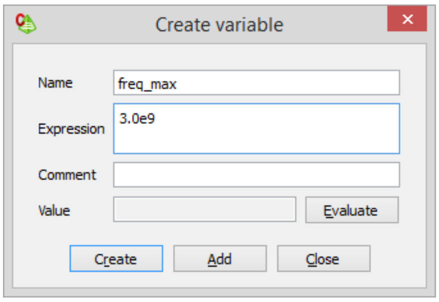
To set the model unit, select the *Construct* tab and click the *Model unit* icon (*Define* group). Select *Millimetres* and close the dialog by clicking the *OK* button. The *Unit* section shown on the status bar in the far lower right corner of the CADFEKO window also opens this dialog.

Define variables



The model tree contains a list of predefined variables. To add variables to the list, right-click the *Variables* heading in the tree and select *Add variable* in the context menu or use the appropriate button from the ribbon (*Construct* tab → *Define* group).

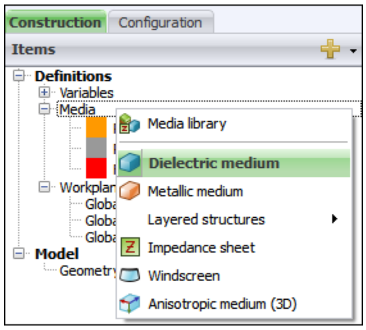
Variables are created by entering a variable name and expression and then clicking the *Add* button. A short comment for the variable can be added to the *Comment* field. The including of comments for variables is optional. Add the following variables:



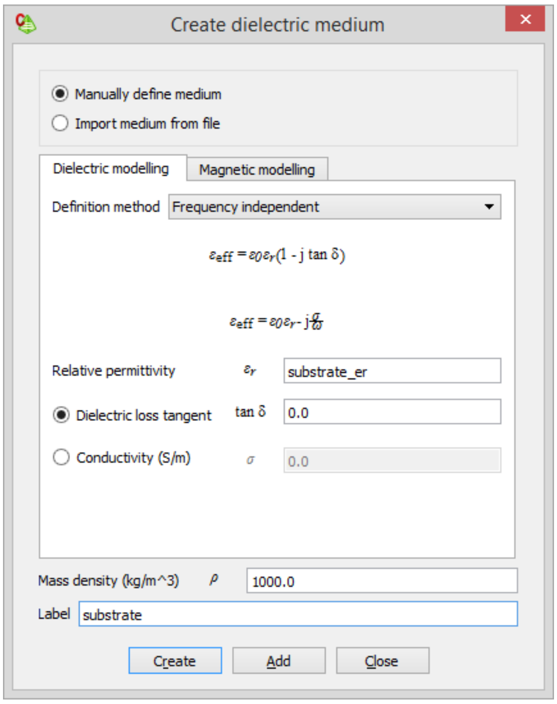
Name	*Expression*
freq_max	3.0e9
freq_min	2.6e9
lambda_min	`c0/freq_max*1000`
patch_d	33.2
patch_w	46.8
substrate_er	2.2
substrate_h	2.87
feed_dist	8.9
feed_rad	1.3/2

After all the variables have been created, click the *Close* button to close the dialog. The comment of a variable is displayed when hovering with the mouse over the variable name (if available).

Define a dielectric medium



 Media can either be defined by selecting the *Construct* tab and clicking the *Media* icon (*Define* group) or by means of the tree. Right-click *Media* in the *Construction* tree and select *Dielectric medium* to create a new dielectric medium definition. All dielectric media must be created before it can be used in the model.

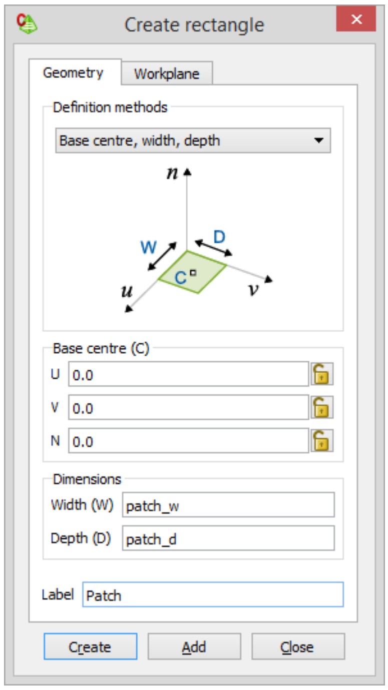


 For the patch, we have selected to simulate the substrate as a lossless material with a relative dielectric constant of 2.2.

Select the *Dielectric modelling* tab and set the *Definition method* to *Frequency independent*.

We have already created a variable for the relative permittivity, so simply enter `substrate_er` in the *Relative permittivity* field. Keep the default value of 0 for the *Dielectric loss tangent* and enter the label name `substrate` into the *Label* field.

Create the patch



 Ensure that the *Construct* tab is selected and click the *Rectangle* icon (*Create surface* group). Under the *Definition methods*, select *Base center, width, depth*.

Enter the dimensions as indicated in the table.

Field	*Value*
Width	`patch_w`
Depth	`patch_d`
Label	`Patch`

Click the *Create* button to create the patch and to close the dialog. Variables and named points can be entered by pressing <Ctrl><Shift> and clicking with the mouse the respective variable or named point in the tree.

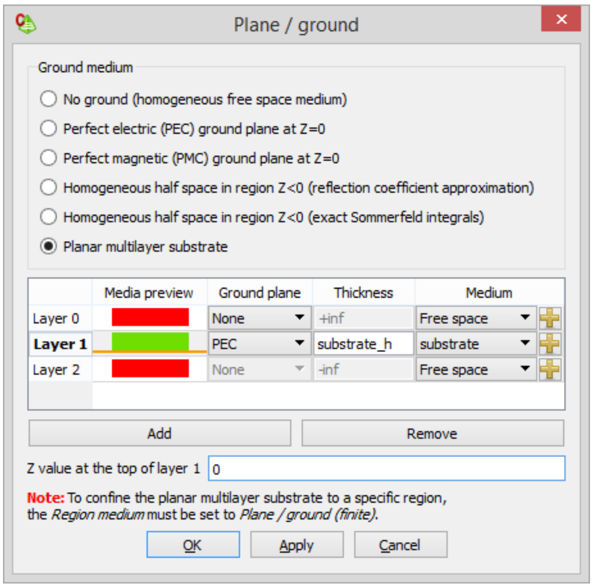
 Click the *Zoom to extents* icon (*View* tab → *Zooming* group) to show the entire geometry. Alternatively, press <F5> on the keyboard.

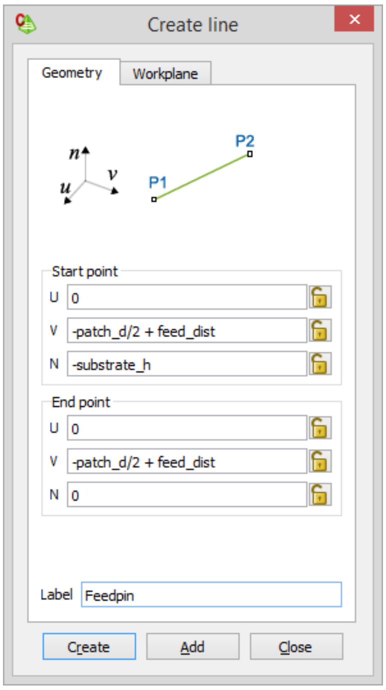
Define the infinite ground plane

Now we add the substrate and the ground plane. Click the *Planes/Arrays* icon on the *Construct* tab of the ribbon (*Structures* group) and select *Plane/ground* from the dropdown menu.

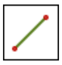
Select *Planar multilayer substrate* as the ground medium. For this example we require only a *PEC* ground plane for *Layer 1*. (These settings are normally already set in the dialog by default.)

For *Layer 1* define a *PEC* ground plane with a thickness of *substrate_h* and of medium *substrate*. Note the *Media preview* on the *Plane / ground* dialog. Click the *OK* button to create the infinite plane and close the dialog.





Create the feed pin



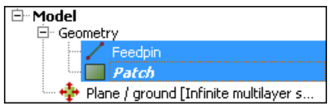
Click the *Line* icon (*Create curve* group) to create the feed pin. We have already created variables for the distance of the feed pin from the edge of the patch. Simply enter the values in the table below.

Corner	*Coordinate*	*Expression*
Start point	U	0
	V	`-patch_d/2 + feed_dist`
	N	`-substrate_h`
End point	U	0
	V	`-patch_d/2 + feed_dist`
	N	0

Enter `Feedpin` in the *Label* field. Click the *Create* button to close the dialog.

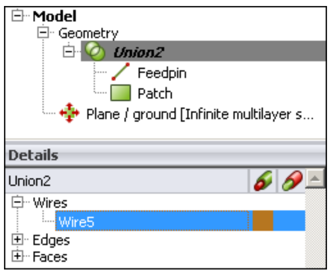
Union the pin and the patch

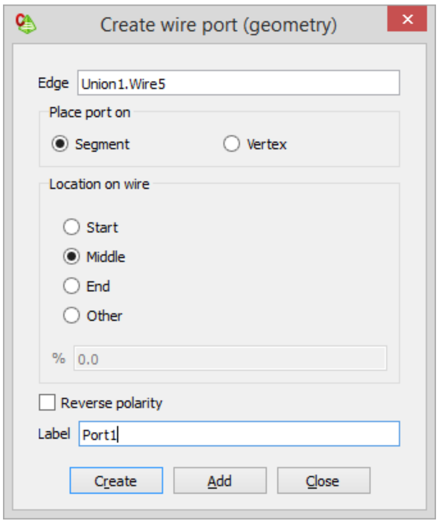
The two geometry parts have to be unioned so that the segment end points coincide with mesh vertices when the mesh is created later. Select the two geometry parts in the tree (using <Ctrl>) and click the *Union* icon (*Modify* group).

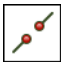


Add a port for the source

After creating the pin, we have to add a port to the feed pin. A port is simply a connection point for loads and sources. Select *Union1* in the tree and its corresponding wire in the details tree.






Select the *Source/Load* tab and click the *Wire port* icon (*Ports* group).

The *Create wire port* dialog is displayed and the edge field is automatically populated with the correct edge. (The name of the *Edge* may differ.) Select *Middle* to place the port in the middle of the wire.

Click the *Create* button to create the wire port and close the dialog.

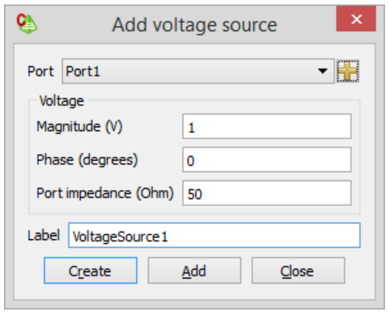
Add a voltage source

We are now going to add a voltage source to the port.

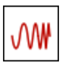

Ensure that the *Source/Load* tab is selected, click the *Voltage source* icon (*Sources on ports*). The values on the *Add voltage source* dialog do not have to be changed.

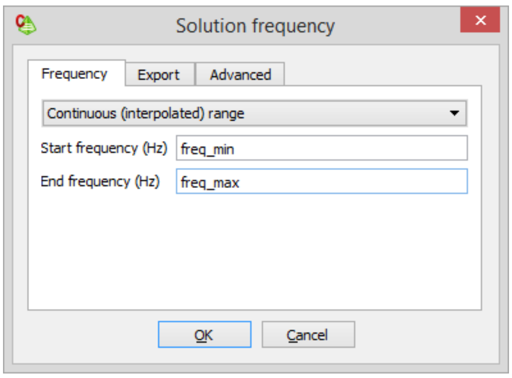
Click the *Create* button to create the voltage source and close the dialog.

Adding the voltage source should change the tab to the *Configuration* tab, as opposed the *Construction* tab where we created the geometry and the port. Sources, loads and the frequency settings are set on the *Configuration* tab.



Set the simulation frequency

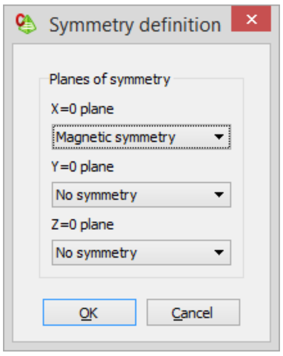

Click the *Frequency* icon (*Settings* group) to set the frequency range of the simulation. Select the *Continuous (interpolated) range* and enter the start and end frequency as listed in the table.



Field	*Value*
Start frequency (Hz)	`freq_min`
End frequency (Hz)	`freq_max`

Also note that variables can be entered by positioning the cursor in the respective edit field and then press <Ctrl> <Shift> and clicking the desired variable.

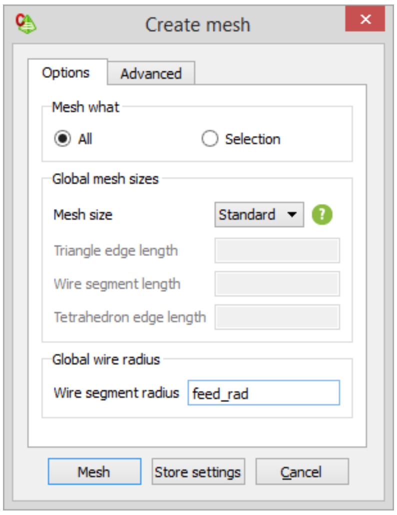
Define model symmetry



The model has magnetic symmetry at X=0. The required resources (memory and time) can be reduced by allowing the solution kernel to utilise symmetry. Note that this step is optional and the user is encouraged to remove symmetry later and to compare the difference in solution time and memory usage.

Select the *Solve/Run* tab and click the *Symmetry* icon (*Solution settings* group). Set *Magnetic symmetry* at the *X=0 plane*. Click the *OK* button to close the dialog.

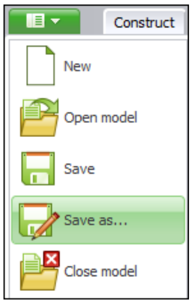
Mesh the model



The model has now been created and all that is left is to mesh the model before we can simulate and view the results.

Select the *Mesh* tab and click the *Create mesh* icon (*Meshing* group) or use the <Ctrl><M> shortcut to open the *Create mesh* dialog.

Set the *Mesh size* to *Standard* and the *Wire segment radius* to `feed_rad`. Click the *Mesh* button to create the mesh and close the dialog.



Save the model by clicking the *Application menu* icon and selecting from the menu *Save as...* and entering a file name. The shortcut key combination to save is <Ctrl><S>.

Validate the model

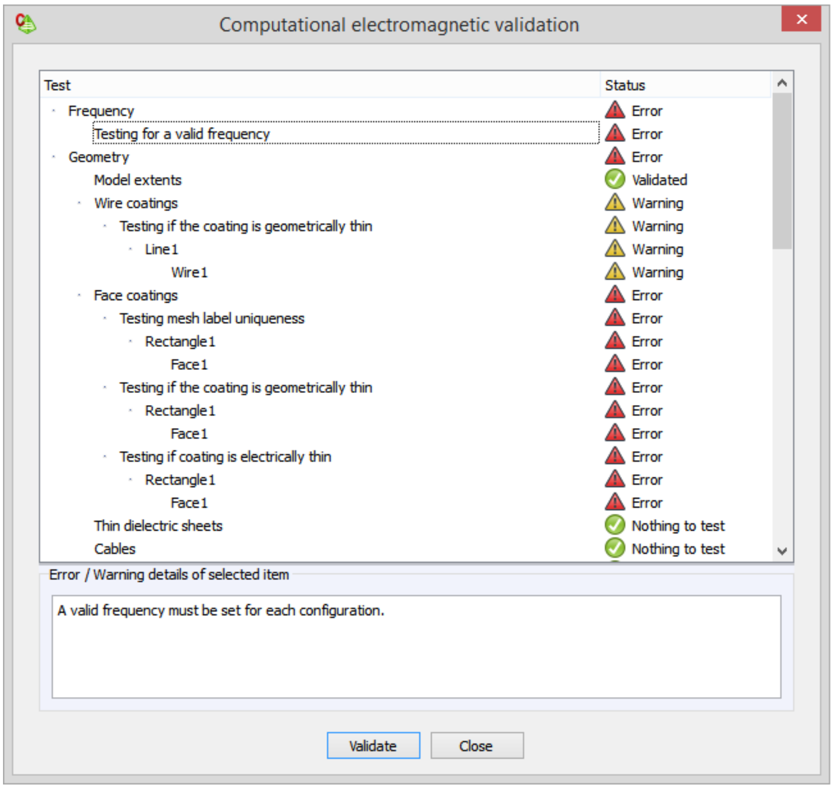
The model should now be computational electromagnetically (CEM) validated. The purpose of this validation is to ensure that errors regarding the *Frequency*, *Geometry*, *Mesh* and *Solution* are found and corrected before the FEKO solver is run.

Run *CEM validate* by selecting the *Solve/Run* tab and clicking the *CEM Validate* icon (*Validate* group).

When any errors are found, an error can be selected resulting in a short description of the error being shown in the *Error/Warning details of the selected item* window.

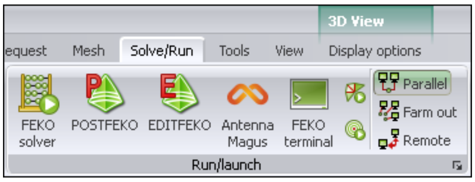
An example of the *CEM validate* dialog is displayed in the image. A number of errors and warnings are indicated to illustrate the feedback that can be expected.

Click the *Close* button to close the dialog.



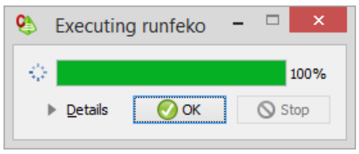
Simulate the model

Run the FEKO solver by selecting the *Solve/Run* tab and clicking the *FEKO solver* icon (*Run/Launch* group).



It can also be launched by using the <Alt><4> shortcut key or by clicking the *FEKO solver* icon on the *Quick access toolbar* which include the application launchers.

The FEKO solver is launched in a window displaying calculation information as well as any warning and error messages. No warning or error messages should be generated if all the steps have been followed correctly. Once the calculation has completed, click the *OK* button to close the message window.



3.3.2 Viewing the results

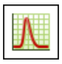
The model creation and simulation process is complete. We can now use POSTFEKO to view the real and imaginary input impedance and the reflection coefficient of the patch antenna. The steps are summarised below.

- Create a Cartesian graph.
- Display the real part of the input impedance over the entire frequency range.
- Add another trace to display the imaginary part of the input impedance on the same graph.

- Change the line colour, line style and line weight of the traces if required.
- Add another Cartesian graph to display the reflection coefficient of the antenna.

Run POSTFEKO by using the <Alt><3> shortcut key, application launchers or from the CADFEKO ribbon. POSTFEKO opens by default with a single 3D window that displays the geometry of the model.

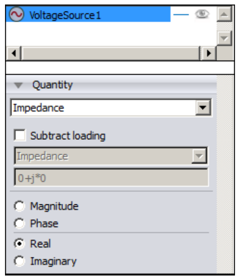
Display real and imaginary source impedance

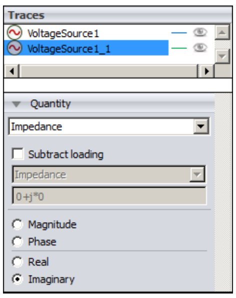


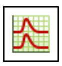
Click the *Cartesian* icon (*Create new display* group on the *Home* tab) to create a blank Cartesian graph where traces can be added.

Click the *Source data* icon (*Home* tab, *Add results* group) and select *VoltageSource1* from the dropdown list.

On the *Result palette* (to the right of the POSTFEKO window), *Traces* section, select the trace *VoltageSource1*. Set the *Quantity* to *Impedance*. Select the *Real* part of the impedance in the *Quantity* section.







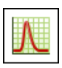
Select the *Trace* tab on the *Cartesian* contextual tab. Click the *Duplicate trace* icon (*Manage* group). A duplicate trace is now created with the label *VoltageSource1_1*. The *Duplicate trace* option is also available on the context menu and <Ctrl><K> keyboard shortcut.

On the *Result palette*, select the second trace (*VoltageSource1_1*) in the *Traces* section. Set the impedance as *Imaginary* in the *Quantity* section.

The graph should now contain two traces. Select one of the two traces by either clicking the trace on the graph or selecting the trace in the *Traces* section in the *Result palette*. A selected trace will be indicated by rectangular selection handles.

On the *Format* tab (*Cartesian* contextual tab), click the *Line colour* icon (*Line* group) and select a colour of your choice from the dropdown menu. Similarly, the *Line style* and *Line weight* may be changed.

Display the reflection coefficient

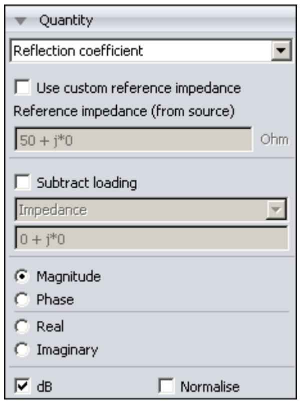


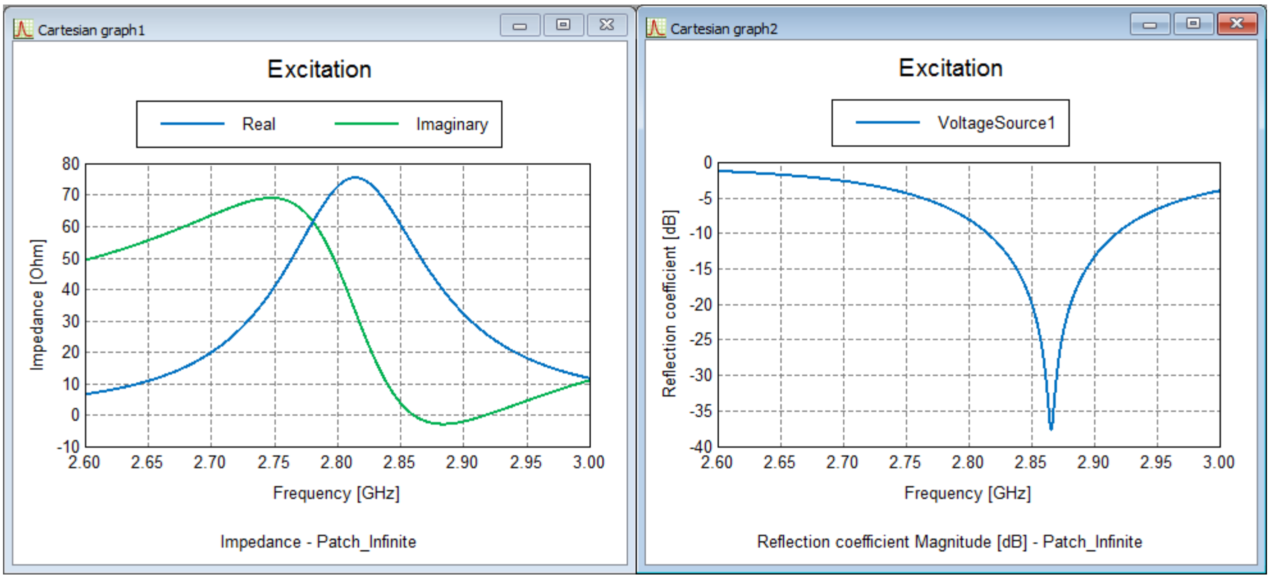
Add another Cartesian graph by selecting the *Home* tab and clicking the *Cartesian* icon. A second graph is now available with the label *Cartesian graph2*.

Click the *Source data* icon (*Home* tab, *Add results* group) and select *VoltageSource1* from the dropdown menu. The reflection coefficient of the patch antenna is displayed on the graph *Cartesian graph2*.

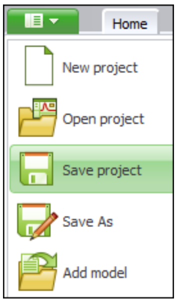
The default display for reflection coefficient is linear, but the reflection coefficient is better viewed on a dB scale.

On the *Result palette* (*Quantity* section), select dB by checking the *dB* checkbox.





Save the model



Save the POSTFEKO session file to ensure the view is maintained for viewing later. The default name and location for the POSTFEKO session file should be used when saving. The session file may be saved by clicking the application menu icon and selecting the *Save* option from the menu.

The patch antenna example with an infinite ground plane has been completed. In the next section we change the model to be more realistic by simulating a patch antenna with a finite size ground plane.

3.4 Patch on finite substrate

The model of the patch is now going to be extended to a patch on a finite substrate. This model is closer to a physical model, but calculation time is increased slightly.

New concepts are introduced such as:

- mesh refinement on edges.
- creating a dielectric region.
- setting specific faces as metallic.

3.4.1 Extending the model

Extending the model in CADFEKO requires the following steps:

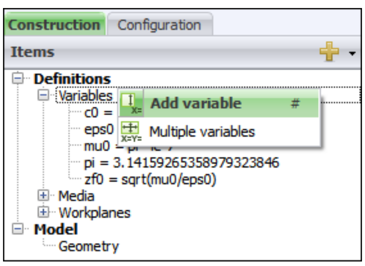
- Remove the previously defined multi-layer substrate.
- Define more variables.
- Create a new substrate with specified dimensions.
- Union the geometry.
- Set the ground and patch face to metallic.
- Mesh the model and validate the resulting mesh.
- Run the FEKO solver.

Leave POSTFEKO open and switch to CADFEKO. We will start by removing the infinite plane from the existing model.

Remove the infinte ground

Select the *Construct* tab and click the *Plane / ground* icon (*Structures* group). On the *Plane / ground* dialog select *No ground (homogeneous free space medium)*. Click the *OK* button to close the dialog. This removes the infinite plane.

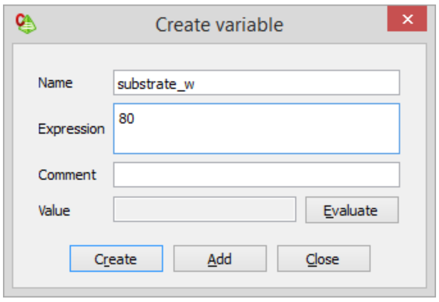
Define variables



Add two new variables to the model. To add the variables to the list, right-click the *Variables* heading in the tree and select *Add variable* in the context menu.

Variables are created by entering a variable name and expression and then clicking the *Create* button. Add the following variables:

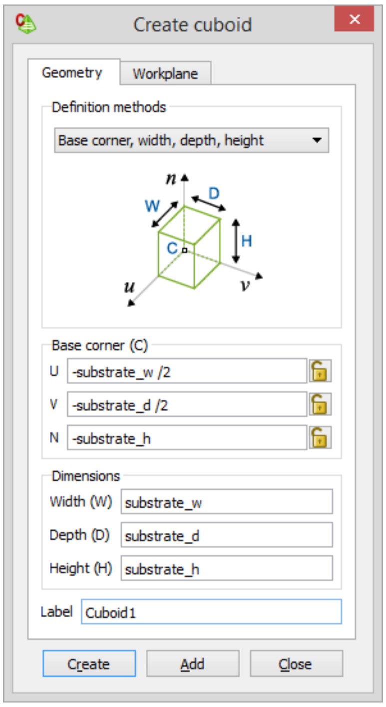
Name	*Expression*
substrate_w	80
substrate_d	50



After all the variables have been created, click the *Close* button to close the dialog.

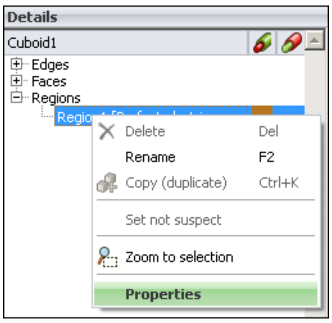
Create finite substrate

Click the *Cuboid* icon (*Construct* tab, *Create solid* group) and fill in the fields as listed in the table. Note that it is not necessary to create variables when defining geometry. The values can also be entered directly, but in this example we have already defined the variables. Variables can also be entered by pressing <Ctrl><Shift> and clicking with the mouse the variable in the tree.

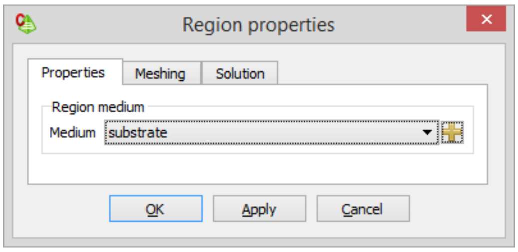


Field	*Value*
U	`-substrate_w/2`
V	`-substrate_d/2`
N	`-substrate_h`
Width (X)	`substrate_w`
Depth (Y)	`substrate_d`
Height (Z)	`substrate_h`

Click the *Create* button to create the cuboid and close the dialog.



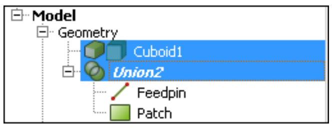
Ensure that the newly created cuboid is selected in the model tree. Right-click the region in the details tree and select *Properties* to open the *Region properties* dialog.



Set the *Region medium* to *substrate*. This is the substrate dielectric medium that we created at the start of the example.

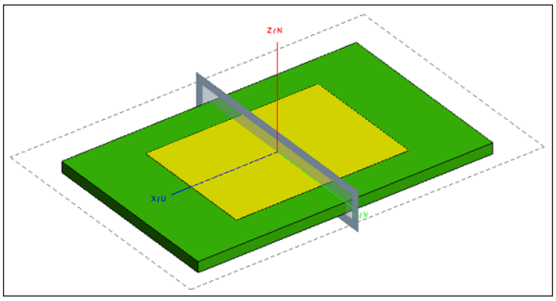
Click the *OK* button to close the dialog.

Union the substrate and the patch



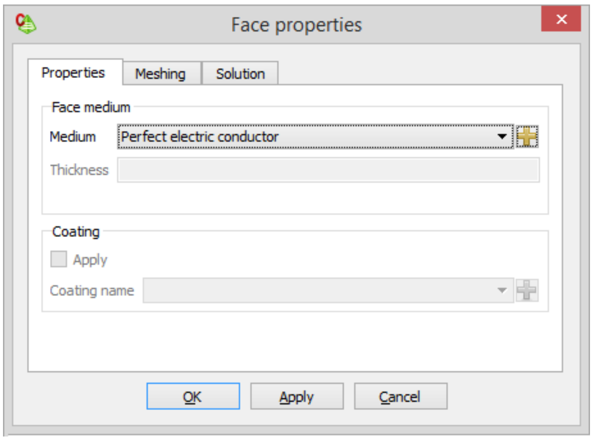
The two geometry parts have to be unioned so that the segment end points coincide with mesh vertices. Select the two geometry parts (using <Ctrl>) and click the *Union* icon.

Set faces to metal



Clicking more than once on the same object cycles through the CADFEKO selection types and thus it is not required to change the selection type manually.

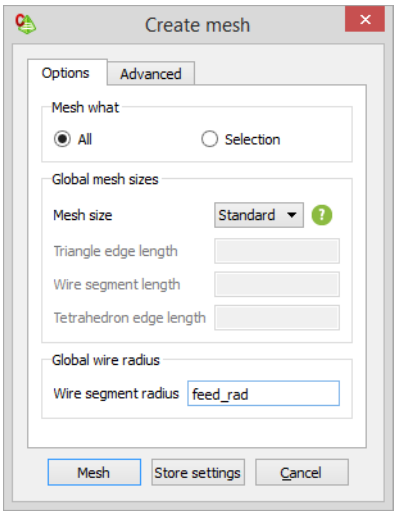
In the 3D view, continue to left-click the face of the patch until it is highlighted in yellow. In the details tree, right-click the highlighted face below the *Faces* group and select *Properties*.



Set the *Face medium* to *Perfect electric conductor* (PEC) and click the *OK* button to close the dialog.

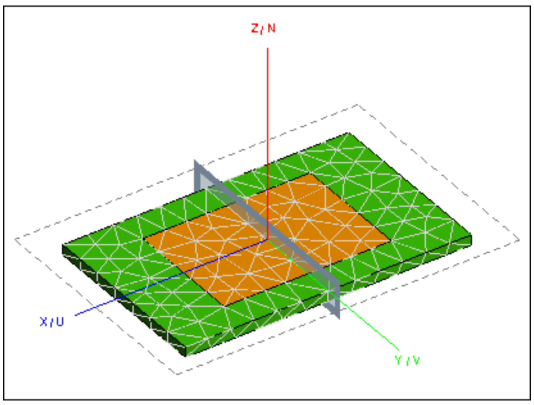
This process of selecting the face and setting it to PEC has to be repeated for the ground plane. Rotate the model so that the bottom of the model is visible. Select the face and set its properties to PEC in the same way as we did for the patch.

Mesh the model



The model has now been created and all that is left is to mesh the geometry before we can simulate and view the results.

Select the *Mesh* tab and click the *Create mesh* icon (*Meshing* group) or use the quick mesh shortcut, <Ctrl><Shift><M>, if the meshing settings have been previously defined.



Before we perform the simulation we should verify that the model that we created is indeed what we wanted to create. CADFEKO has several model validation tools and we are only going to use the mesh colouring tools in this example.

The meshed patch should look similar to the image (the medium colour will probably be different).

Save, validate and simulate

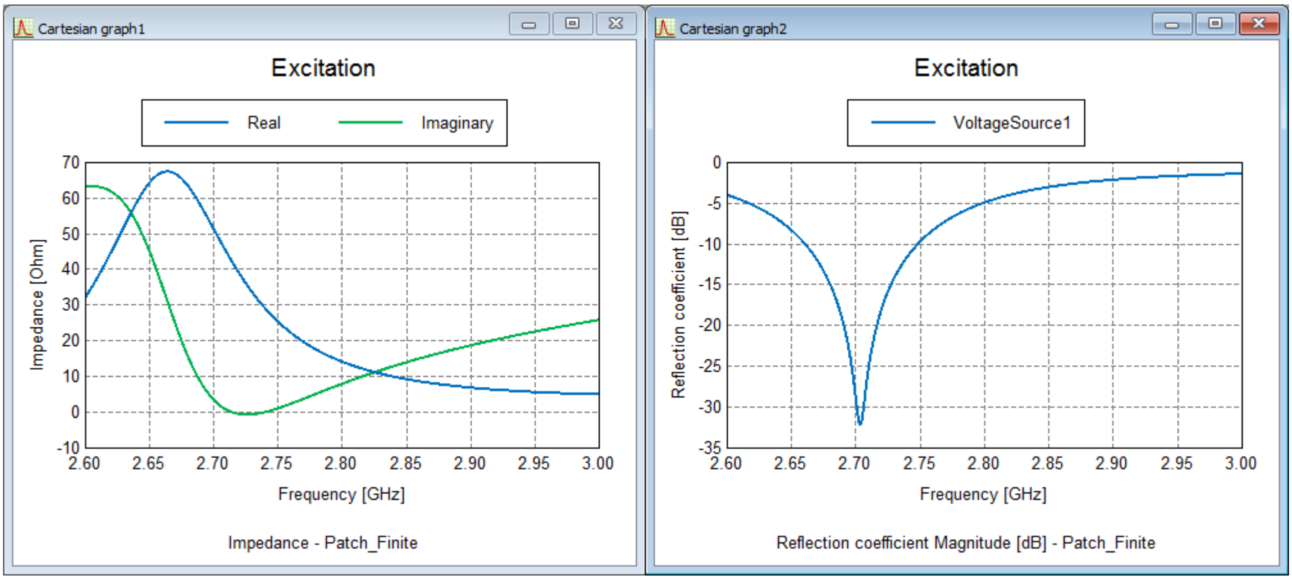
Save the model. The key combination (shortcut) to save is <Ctrl><S>. Remember to save often when creating models to prevent work being lost due to system failures.

Run the *CEM validate* by selecting *Solve/Run* tab and clicking the *CEM validate* icon (*Validate* group).

Run the FEKO solver by selecting the *Solve/Run* tab and clicking the *FEKO solver* icon or using the <Alt><4> key combination. A message window asking if the results should be replaced may appear, click *Yes* to allow the solution kernel to calculate the new results. The FEKO solver is launched in a window displaying calculation information as well as any warning and error messages. No warning or error messages should be generated if all the steps have been followed correctly. Once the calculation has completed, click the *OK* button to close the message window.

3.4.2 Viewing the new results

The results of the simulation can be viewed in POSTFEKO after completing the simulation. Since we have already created a POSTFEKO session for this project, all display windows are visible without any changes. POSTFEKO automatically detects if the FEKO solver results have changed and updates the session data.



3.5 Closing remarks

This example has demonstrated two ways to simulate a patch antenna excited with a pin such as an SMA connector. Many concepts have been introduced in this simple example that are applicable to models commonly created in CADFEKO.

The important factors to remember is the difference in solution time and the deviation in the results. The model with an infinite substrate required fewer triangles to model and simulated faster than the model with a specified substrate size. The difference in the resonance frequency is approximately 6.5% and would become greater if we were to decrease the substrate dimensions further. It is important to always consider how well your model represents real life. The correlation between the results of the two models should improve if the size of the finite substrate is increased to better represent the infinite approximation.

4 Getting started project: EMC coupling

4.1 Example overview

This example considers the coupling between a typical monopole antenna and a loaded transmission line as shown in Figure 4-1. Both the antenna and the transmission line consist of wire conductors (we refer to structures as wires when they are conducting bodies whose lengths are significantly longer than their diameters).

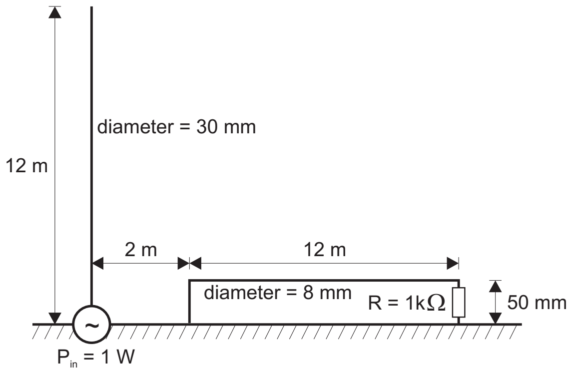


Figure 4-1: Sketch of the model.

4.2 Before starting the example

Before starting this example, please ensure that the system satisfies the minimum requirements before starting. A user should also ascertain whether the topics presented in this example are relevant to the intended application and FEKO experience level.

The topics demonstrated in this example are:

- Using a perfectly conducting infinite ground plane.
- Loading of a wire with a complex impedance.
- Adding a voltage source to a wire.
- Using adaptive frequency sampling to obtain continuous data.
- Viewing the simulated input impedance and currents in POSTFEKO.

The requirements for this example are listed below.

- FEKO 2017.2 RELEASE or later should be installed[a] with a valid licence.
- It is recommended that the demo video be watched before attempting this example.
- This example should not take longer than 70 minutes to complete.

[a]See the FEKO Installation Guide to install Altair FEKO.

The models referred to in this example can be found in the

```
examples/GetStarted_models/Project4-EMC_Coupling
```

directory of the Altair FEKO installation or downloaded from our website.

4.3 Creation of the geometry in CADFEKO

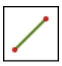
The first step in every FEKO solution is to construct the model. Start by launching CADFEKO which opens with the start page. Click the *Create a new model* icon.

Save the model in a new directory where it can later be found for further experimentation. CADFEKO saves its models as `*.cfx` files.

The model creation steps that we are going to perform can be summarised as:

- The monopole is created as a single line element with a local wire radius.
- The transmission line is created as a polyline element.
- The ground plane is defined using an infinite reflection ground.
- Add a port and voltage source on the monopole.
- Set the radiated power of the model.
- Add a port and a complex load to the transmission line.
- Set the solution frequency.
- Mesh the model and run the FEKO solver.

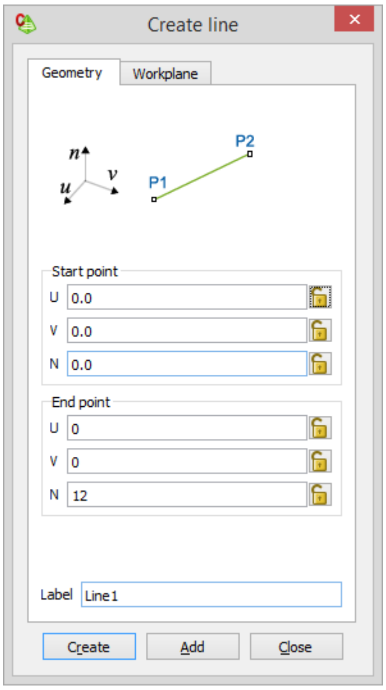
Create the monopole

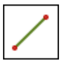


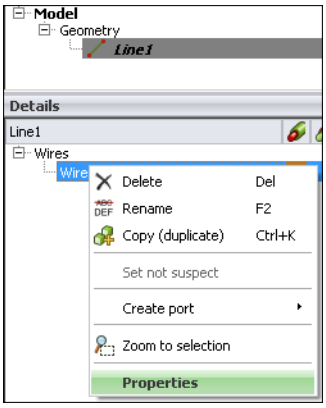
The first step is to draw the monopole along the positive Z axis. Select the *Construct* tab and click the *Line* icon (*Create curve* group). A single line 12 m long is created along the Z axis.

After creating this part, select the *View* tab and click the *Zoom to extents* icon (*Zooming* group) or press the <F5> shortcut key. Note that it may be difficult to see the monopole as a result of the overlap with the Z axis.

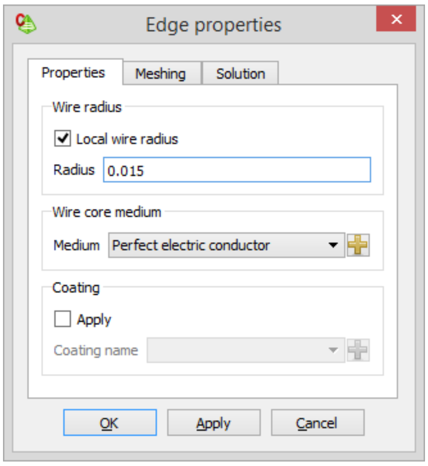
The display of the global axis can be disabled by selecting the *Display options* on the *3D View* context. Click the *Main axes* icon (*Axes* group) to toggle the display on and off for the main axes.





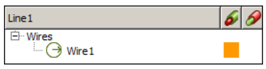


Because the monopole and transmission line wires have different radii, the wire radius for this model cannot be set globally during meshing. The global setting will be set for the transmission line, so a local radius needs to be specified for the monopole. The wire radius is set on the *Edge properties* dialog. To open this dialog, select the line in the tree, expand the *Wires* in the details tree and select the wire. Select the *Properties* option from the context menu.

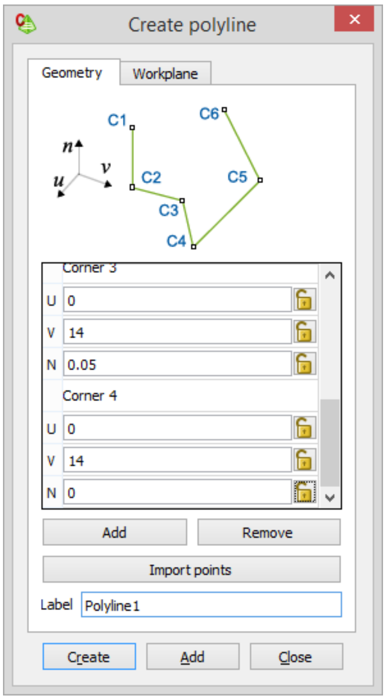


The wire radius is set by checking the *Local wire radius* checkbox and entering 0.015 in the available field.

The icon next to the edge will change in the detail tree, indicating a locally specified radius.



Create the transmission line



The transmission line is created using a polyline curve. To create it, select the *Construct* tab and click the *Polyline* icon (*Create curve* group). By default, two corners are shown. Click the *Add* button (next to the *Remove* button) twice to add the additional corners.

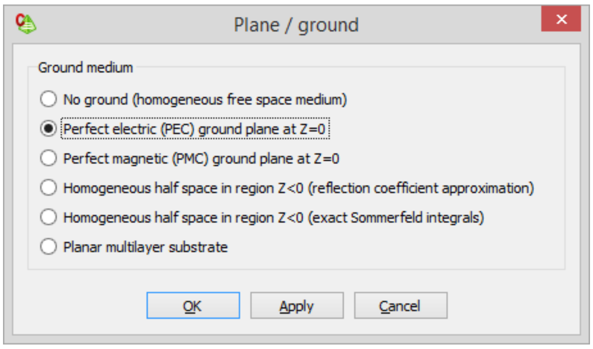
The transmission line is created along the Y axis by defining the specified corners.

	U	*V*	*N*
Corner 1	0	2	0
Corner 2	0	2	0.05
Corner 3	0	14	0.05
Corner 4	0	14	0

Select the *View* tab and click the *Zoom to extents* icon (*Zooming* group) to get the structure to fit inside the 3D view (or press <F5>).

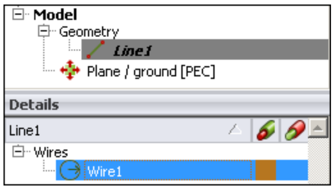
Define the infinite ground

To create the infinite ground plane, a *Perfect electric* (PEC) ground plane at Z=0, is added. To do this, select the *Planes/Arrays* menu button (*Structures* group).

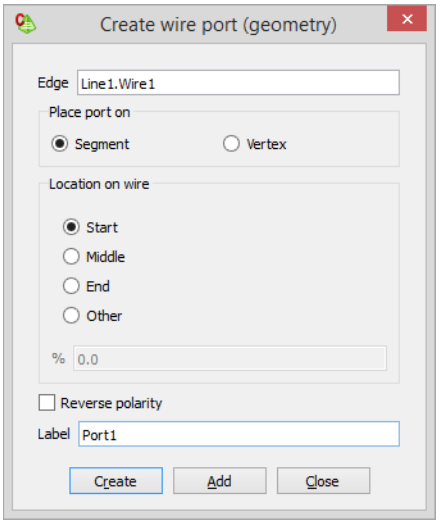


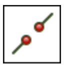
Click *Perfect electric (PEC) ground plane at Z=0.* This method of modelling large ground planes is an efficient compared to discretising a finite sized ground plane.

Define the ports



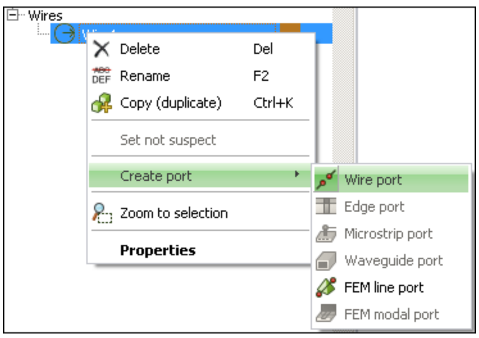
The next step is to define the ports, sources and loads for the model. The first port will be created on the monopole for the source. To create this port, select the *Line1* part in the tree and then select the wire element (*Wire1*) in the details tree that is associated with *Line1*.



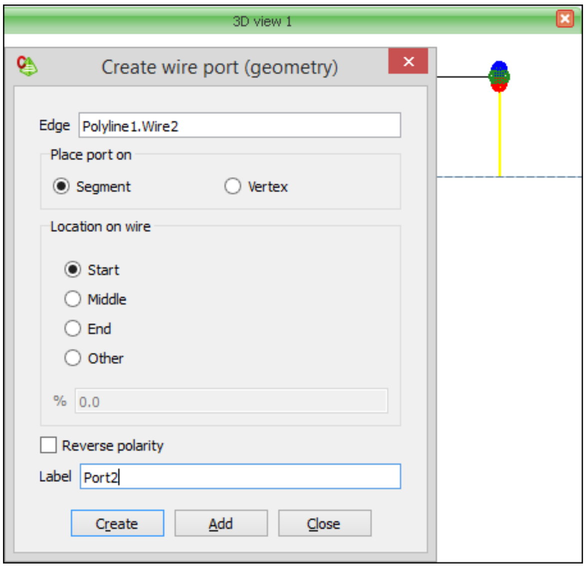


Select the *Source/Load* tab and click the *Wire port* icon (*Ports* group).

The default port settings (start of line, segment port) are used for this model. Click the *Create* button to create the port and close the dialog.



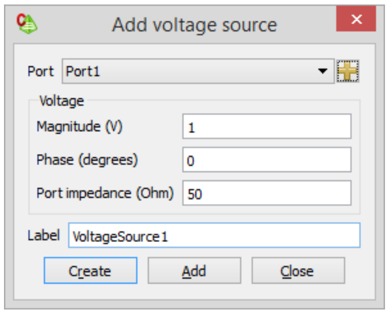
Ports may also be added via the details tree. The port for the load on the transmission line will now be defined. First select the correct edge (the vertical wire farthest from the monopole) in the details tree, right-click and select *Create port* in the context menu. Select *Wire port* from the context menu.



Care needs to be taken to ensure that the correct edge is selected. (The 3D view shows a preview of the port. It is indicated by a sphere with its centre in green.) Again, the default settings can be used for the port. Click the *Create* button to create the port and close the dialog.

Add a source

A Voltage source is now applied to the first port. Select the *Source/Load* tab and click the *Voltage source* icon (*Sources on ports* group).

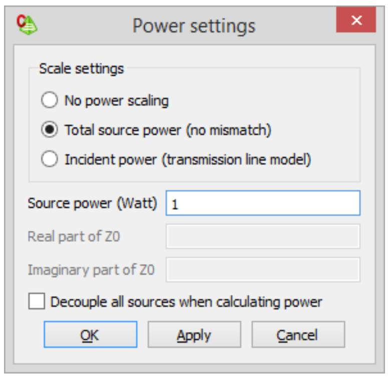


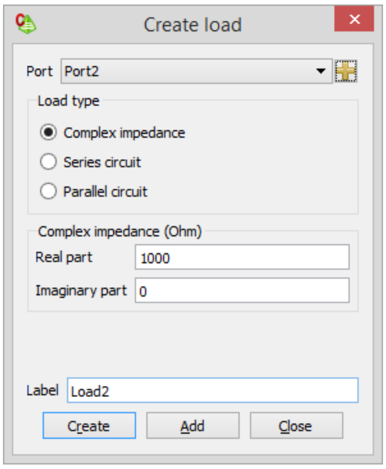
The radiated power must be 1 Watt for this example, but since the input impedance for the monopole is not known, this can not be set by changing the voltage. In the next step, the power settings are changed to scale the radiated power. Therefore leave the default voltage settings. Click the *Create* button to define the voltage source and close the dialog.

Set the radiated power level

Once the source has been created, CADFEKO will change the tree tab to the *Configuration* tab where sources, loads and power settings can be modified.

To set the radiated power for the model, click the *Power* icon (*Settings* group). In this case the radiated power should be 1 Watt, so power losses as a result of source mismatch are deducted before the 1 Watt is calculated. Select the *Total source power (no mismatch)* option. Enter a source power of 1 Watt and click the *OK* button.

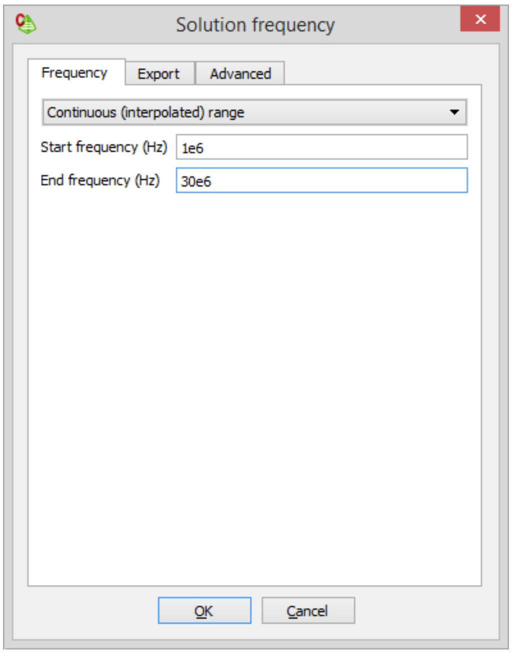




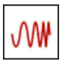
Add a complex load

The resistive load must be added to the second port. Select the *Source/Load* tab and click the *Add load* icon (*Loads/networks* group). Change the port for the load to *Port2* and the real part of the impedance to 1000 Ω. Click the *Create* button to create the load and close the dialog.

The remaining step in the process is to specify the frequency range of interest.

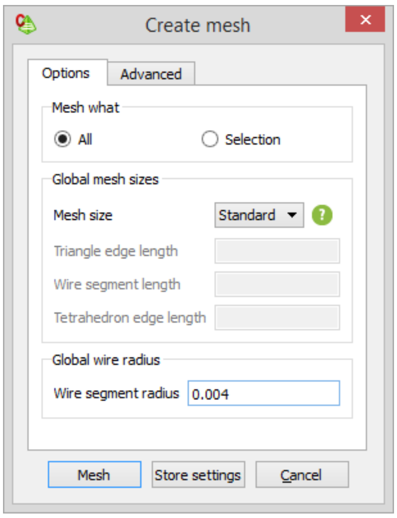


Set the simulation frequency



To set the frequency range for the analysis, select the *Source/Load* tab and click the *Frequency* icon (*Settings* group). For this example, continuous frequency sampling is used where FEKO automatically determines the frequency sampling for optimal interpolation. Select the *Continuous (interpolated) range* option and enter the provided start and end frequencies.

	Frequency
Start frequency	1 MHz
End frequency	30 MHz



Mesh the model

Now the model must be meshed. To create the mesh, select the *Mesh* tab and click the *Create mesh* icon (*Meshing* group) or use the shortcut key <Ctrl><M>.

Set the *Mesh size* to *Standard*. The global *Wire segment radius* is set to that of the transmission line, which is 4 mm. As the model unit is in metre, enter a value of 0.004. (The radius of the monopole was set locally, and local sizes overwrite global sizes.)

4.4 CEM validation

The model should now be computational electromagnetically validated. The purpose of this validation is to ensure that errors regarding the *Frequency*, *Geometry*, *Mesh* and *Solution* are found and corrected before the FEKO solver is run.

Run the *CEM validate* by selecting the *Solve/Run* tab and clicking the *CEM validate* icon (*Validate* group).

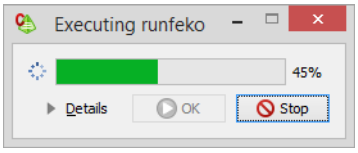
When any errors are found, the error can be selected resulting in a short description of the error being shown in the *Error/Warning details of the selected item* window.

4.5 Obtaining a solution

After completing the model preparation, the solver should be invoked to calculate the results. No requests were added to this model since impedance current information are automatically calculated for all voltage and current sources. A prompt will appear asking whether the model should be saved before the solver executes.

Run the FEKO solver by selecting the *Solve/Run* tab and clicking the *FEKO solver* icon (*Run/Launch* group).

It can also be launched by clicking the *FEKO* solver icon on the *Application launchers* at the top right of the CADFEKO window. A window will open, giving step by step feedback as the simulation progresses. The solver will use approximately 39 frequency samples in the band. (Note that this value can vary slightly.)

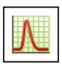


4.6 Visualisation of results

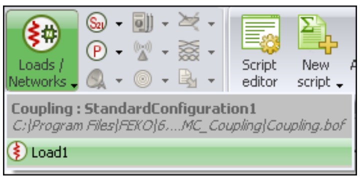
The model creation and simulation process is complete. We can now use POSTFEKO to view the results.

Ensure that CADFEKO is the active application. Run POSTFEKO by using the <Alt><3> shortcut key, application launchers or from the CADFEKO ribbon. POSTFEKO opens by default with a single 3D window that displays the geometry of the model.

Display the load current

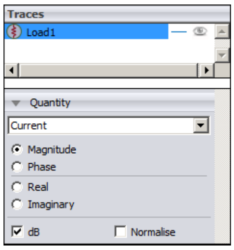


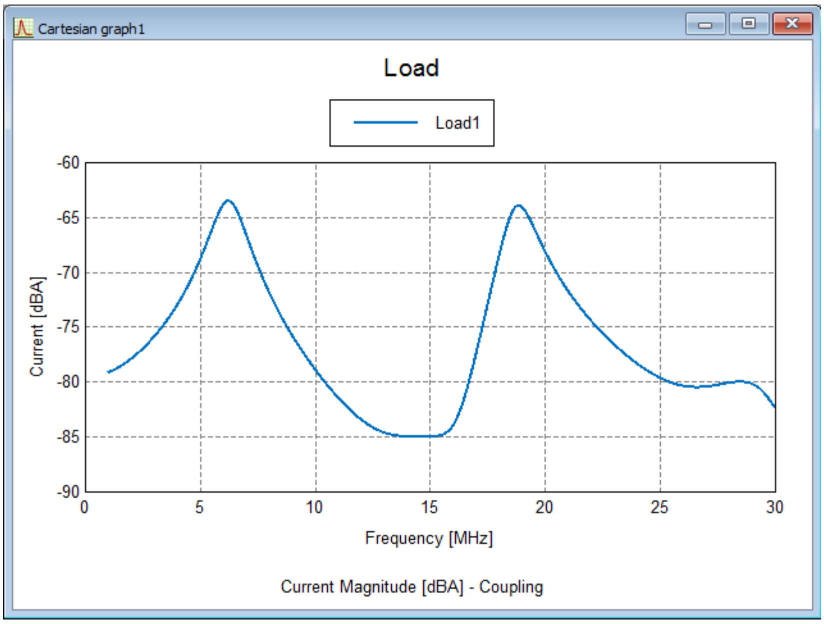
To show a graph of the current in the load on the transmission line, click the *Cartesian* icon (*Home* tab, *Create new display* group) on the ribbon.



Click the *Loads/networks* icon (*Home* tab, *Add results* group) and from the dropdown menu select *Load1*.

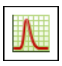
Ensure that the trace *Load1* is selected in the *Result palette*, *Traces* section. Select the dB checkbox to change the scaling to dB in the *Quantity* section.





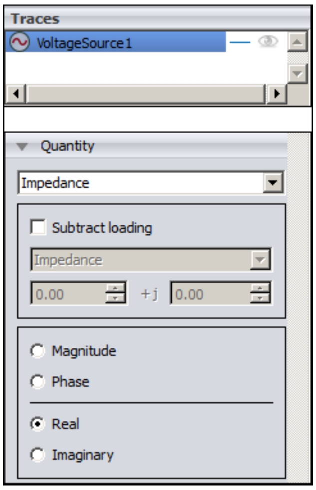
Display the source input impedance

We are also going to look at the input of the source on a new Cartesian graph.

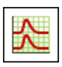


Click the *Cartesian* icon (*Home* tab, *Create new display* group) to create a new Cartesian graph.

Click the *Source data* icon (*Home* tab, *Add results* group) and select *VoltageSource1* from the dropdown list.



Ensure that the trace *VoltageSource1* is selected. In the *Quantity* section, select *Impedance* as quantity. Also select the *Real* real part of the impedance.

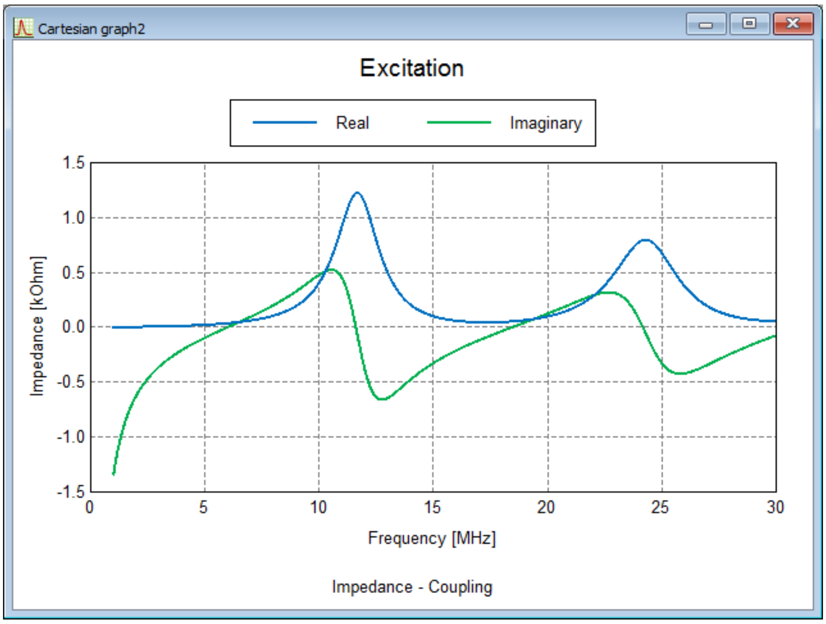


Select the *Trace* tab on the *Cartesian* contextual tab. Click the *Duplicate trace* icon (*Manage* group). A duplicate trace is now created with the label *VoltageSource1_1*. The *Duplicate trace* option is also available on the context menu and <Ctrl><K> keyboard shortcut.

Select the trace *VoltageSource1_1* and for the impedance select *Imaginary*.

Select one of the two traces by either clicking the trace on the graph or selecting the trace in the *Result palette, Traces* section. A selected trace will be indicated by rectangular selection handles.

On the *Format* tab (*Cartesian* contextual tab), click the *Line colour* icon (*Line* group) and select a colour of your choice from the dropdown menu. Similarly the *Line style* and *Line weight* may be changed.



4.7 Closing remarks

This example has shown the construction, configuration and solution of an EMC coupling problem. The problem description is a monopole antenna and transmission line on an infinite perfectly electric conducting ground plane. Coupling of current into the transmission line is shown from 1 MHz to 30 MHz.

5 Getting started: A waveguide power divider

5.1 Example overview

In this example the transmission and reflection coefficients of a waveguide power divider is calculated at 9 GHz. The power divider has been designed to split the power equally between the two output ports while minimising any power reflected back to the source port. The power is split by placing a metal pin at the junction between the three ports. The waveguide geometry (and instantaneous near field) is illustrated in Figure 5-1.

The model utilises symmetry to reduce memory requirements and to increase the calculation speed.

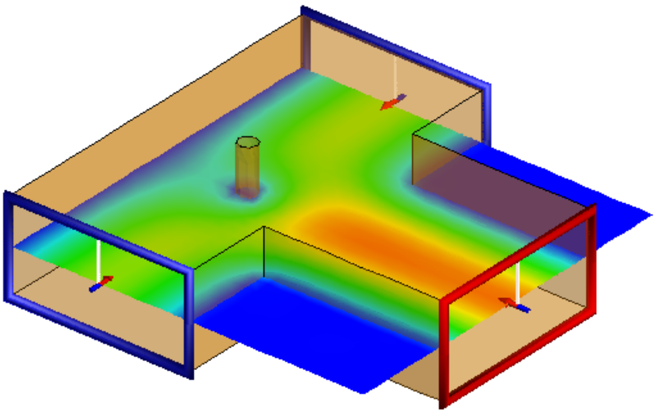


Figure 5-1: Illustration of the waveguide power divider.

5.2 Before starting the example

Before starting this example, please ensure that the system satisfies the minimum requirements before starting. A user should also ascertain whether the topics presented in this example are relevant to the intended application and FEKO experience level.

The topics demonstrated in this example are:

- Using waveguide ports.
- Defining geometry and solution parameters in CADFEKO.
- Utilising symmetry to reduce the required resources (time and memory).
- Running the FEKO solution kernel.
- Viewing the results in POSTFEKO.

The requirements for this example are listed below.

- FEKO 2017.2 RELEASE or later should be installed[a] with a valid licence.
- It is recommended that the demo video be watched before attempting this example.
- Working through this example should not require more than about 30 minutes.

[a] See the FEKO Installation Guide to install Altair FEKO.

While working through this example, the steps should be followed sequentially, otherwise explanations may seem to be out of context.

The models referred to in this example can be found in the

```
examples/GetStarted_models/Project5-Waveguide_Power_Divider
```

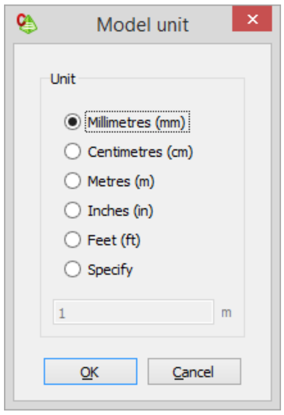
directory of the Altair FEKO installation or downloaded from our website.

5.3 Creation of the model in CADFEKO

The first step in every FEKO solution is to construct the model. Start by launching CADFEKO which opens with the start page. Click *Create a new model*. The model is created using CADFEKO and stored in the `*.cfx` file.

The steps that we are going to follow to create the model are:

- Set the model unit to millimetres.
- Define variables to parametrise the model.
- Create the power dividing pin.
- Create the waveguide sections.
- Move the geometry so that symmetry can be utilised.
- Set the symmetry planes to use.
- Add waveguide ports to the relevant faces.
- Add a source.
- Set the frequency of the solution.
- Add a near field request.



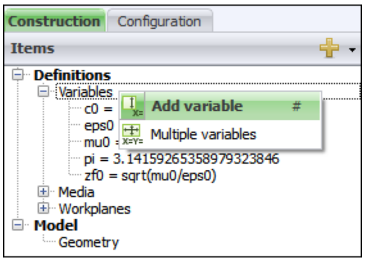
Set model unit

The default unit length in CADFEKO is metres. Since the structure that we are going to build is small, the model unit should be set to millimetres. All dimensions will then be entered in the new model unit.

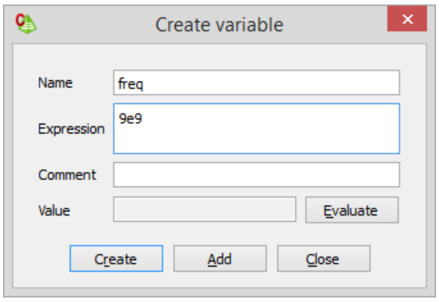
To change the model unit, select the *Construct* tab and click the *Model unit* icon (*Define* group). Select *Millimetres* and close the dialog by clicking the *OK* button. The *Unit* section shown on the status bar in the far lower right corner of the CADFEKO window also opens this dialog.

Define variables

Variables may be defined by selecting the *Construct* tab and clicking the *Variable* icon (*Define* group).



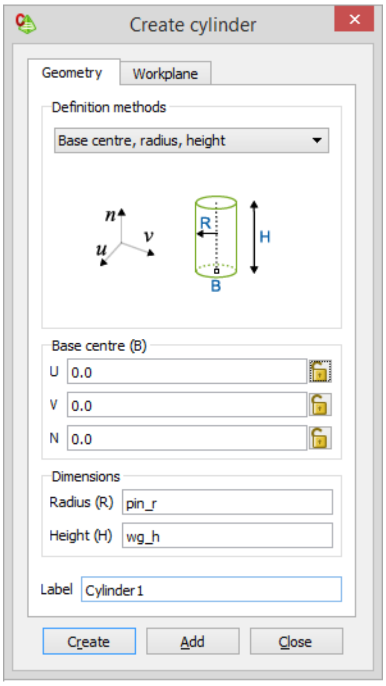
Variables may also be defined via the tree. Right-click the *Variables* group in the model tree (*Construct* or *Configuration* tab) and select *Add variable*. A short comment for a variable can be added to the *Comment* field. The inclusion of comments for variables is optional. The comment of the variable is displayed when hovering the mouse over the variable name (if available).



Create the following variables:

Name	*Expression*
freq	9e9
lambda	`c0/freq*1000`
pin_r	1
wg_h	10
wg_w	20

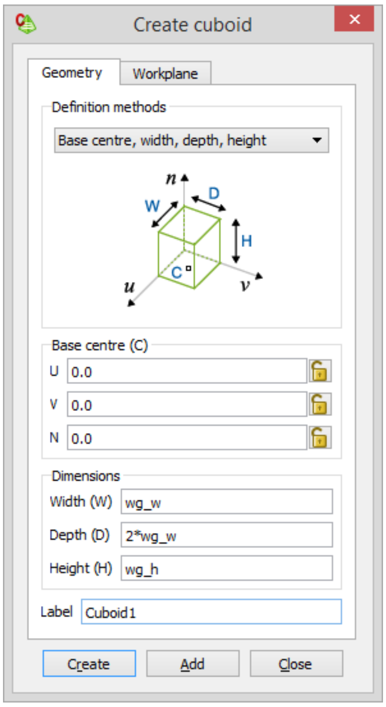
Create cylinder (power dividing pin)



A cylinder will now be created. Select the *Construct* tab and click the *Cylinder* icon (*Create solid* group). The dimensions and position for the cuboid are listed in the table below.

Group	*Field*	*Expression*
Base centre	U	0
	V	0
	N	0
Dimensions	Radius (R)	`pin_r`
	Height(H)	`wg_h`

The variables, `pin_r` and `wg_h`, can be added by holding down <Ctrl><Shift> and clicking the respective variables in the tree. Leave the name *Cylinder1* (default name) in the *Label* field. Click the *Create* button to create the cylinder and close the dialog. Click the *Zoom to extents* icon to fit the 3D display around the geometry - alternatively press <F5>.



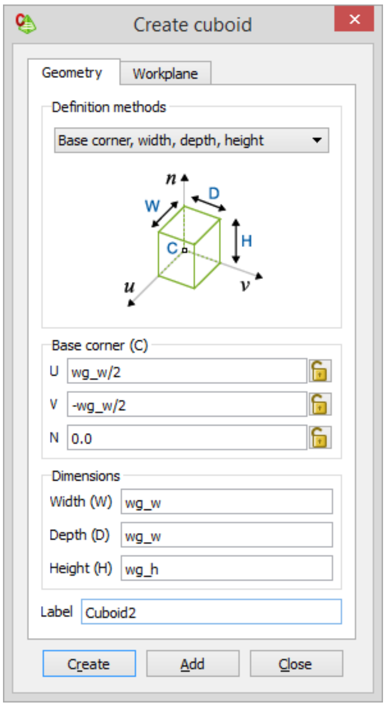
Create cuboids (waveguide sections)

Select the *Construct* tab and click the *Cuboid* icon (*Create solid*).

Create a cuboid with the *Base centre, width, depth, height* definition method. The dimensions and position are listed below.

Group	*Field*	*Expression*
Base centre	U	0
	V	0
	N	0
Dimensions	Width (W)	`wg_w`
	Depth (D)	`2*wg_w`
	Height(H)	`wg_h`

Leave the name *Cuboid1* (default name) in the *Label* field. Click the *Add* button to create the cuboid, but leave the dialog open.

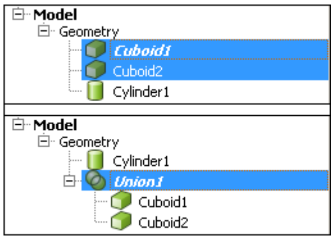


Create a second cuboid with the dimensions and position listed in the table below. Note that the *Base corner, width, depth, height* definition method is specified and not the base centre as with *Cuboid1*.

Group	*Field*	*Expression*
Position	U	wg_w/2
	V	-wg_w/2
	N	0
Dimensions	Width (W)	wg_w
	Depth (D)	wg_w
	Height(H)	wg_h

Leave the name *Cuboid2* (default name) in the *Label* field. Click the *Create* button to create the cuboid.

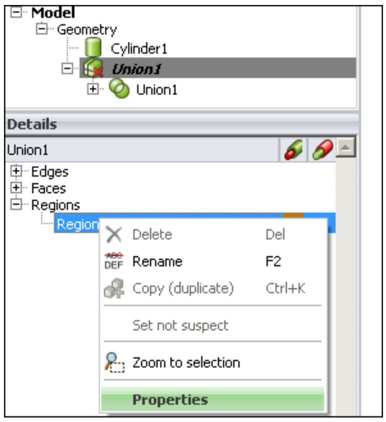
Union and simplify cuboids



The cuboids (*Cuboid1* and *Cuboid2*) must be unioned in order to become a single geometry part. Select *Cuboid1* and Cuboid2 in the model tree using the <Ctrl> key. Select the *Construct* tab and click the *Union* icon (*Modify* group).

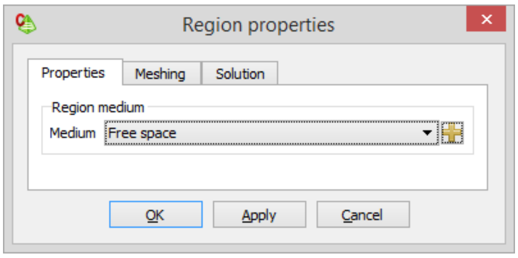
The union of the two cuboids (*Union1*) will now be simplified. Select *Union1* in the model tree and select the *Transform* tab. Click the *Simplify* icon (*Simplify* group) to launch dialog with simplification options. Leave the default settings as is and click the *Create* button. The simplification process removes items that are redundant; in this example the face between the two cuboids (where they touch) is removed.

Set waveguide to be hollow



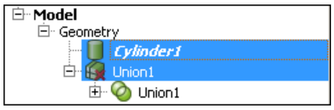
The waveguide section that has been constructed is currently still solid metal, but needs to be hollow with metal side walls. We need to set the region inside the waveguide to be *Free space*.

To set the properties of the interior region of the model, select the part (*Union1*) in the model tree. The details tree now shows a *Regions* branch. Expand this branch to see a list of available regions. In this geometry part only one region (the inside of the waveguide) exists. Right-click this region, and select *Properties*.



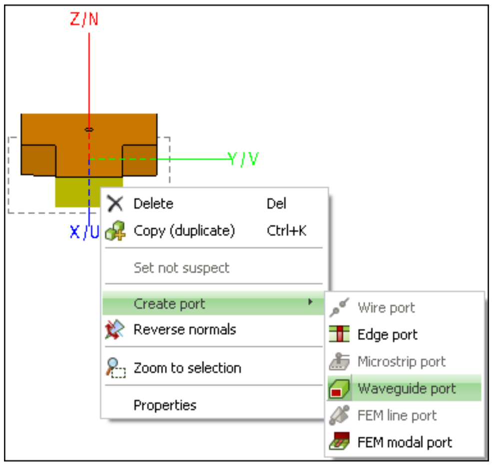
In the region properties dialog, set the *Region medium* to *Free space*. Click the *OK* button to accept and close.

Union waveguide and power dividing pin



The waveguide sections (*Union1*) and the cylinder (*Cylinder1*) must be unioned to become a single geometry part. Select *Union1* and *Cylinder1* in the model tree. Select the *Construct* tab and click the *Union* icon (*Modify* group).

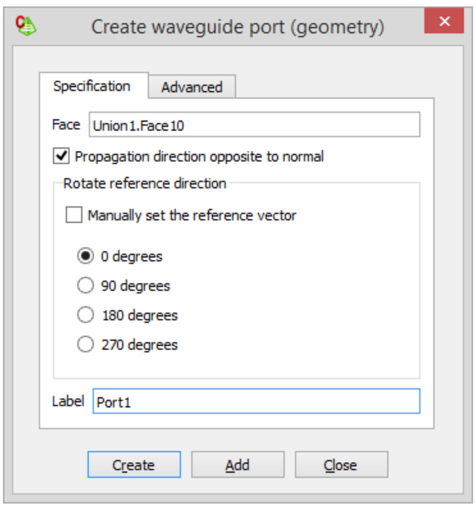
Add wavguide ports



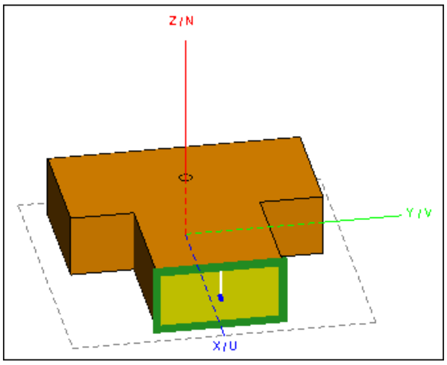
We have created the geometry and we are now going to add the waveguide ports and the source.

In the 3D view, continue to left-click the face where *Port1* should be placed (the face at the maximum X position) until the required face is highlighted. Open the context menu (right-click) of the face and selecting *Create port→Waveguide port* (also available under the *Source/Load* tab in the *Ports* group. Use the default label of *Port1*.

Note that ports are simply connection points on the geometry where sources can be added. Waveguide ports that do not have sources are considered to be absorbing waveguide terminations.



The *Create waveguide port* dialog is displayed. Note that the correct face label is already entered in the *Face* field due to first selecting the face in the 3D view.



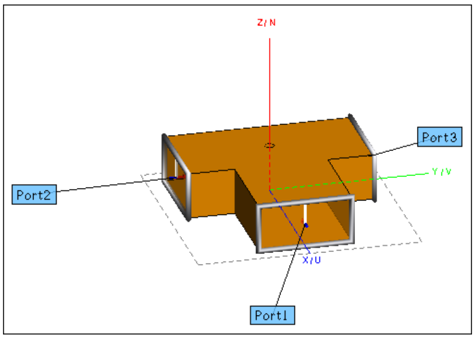
Whilst the *Create waveguide port (geometry)* dialog is open, a preview of the waveguide port will be indicated in green in the 3D window. Note that the reference vector is a silver line on the port face from the centre going in the positive Z direction.

The default label for the port (*Port1*) should be used for this example. Click the *Add* button, but do not close the dialog.

To add the second port, click the *Face* edit box (it has a yellow background colour when it is selected) and then select the face where the second port should be placed (the face at the most negative Y position or the geometry). The default port label, *Port2*, can be used. Click the *Add* button to create the waveguide port.

The third port is added in the same way as the other ports. The correct face for *Port3* is the face at the most positive Y position in the geometry. Select the *180 degrees* option to ensure the correct reference direction for the third port.

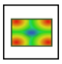
Click the *Create* button to define the final waveguide port and close the dialog.



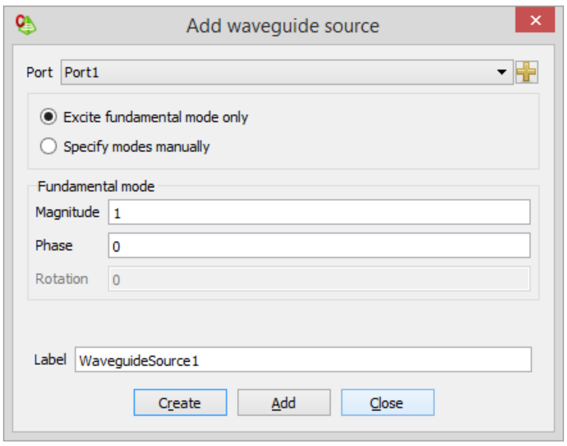
If all the steps have been followed correctly, the structure should look similar to the image. Note the port locations.

It may be necessary to enable the annotations by clicking the *Port annotations* icon in the *Entity display* group (*Display options* tab of the *3D View* contextual tab group).

Add waveguide source to port 1

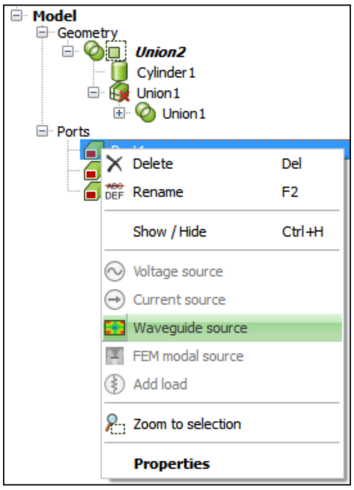


The waveguide source can now be added to port 1. Select the *Source/Load* tab and click the *Waveguide source* icon (*Sources on ports* group). Select *Port1* from the dropdown menu.



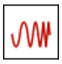
All the default values are used in this example. The fundamental mode for this source will be excited (TE_{10}). It is also possible to add multiple modes as a single source by choosing the *Specify modes manually* option.

Click the *Create* button.

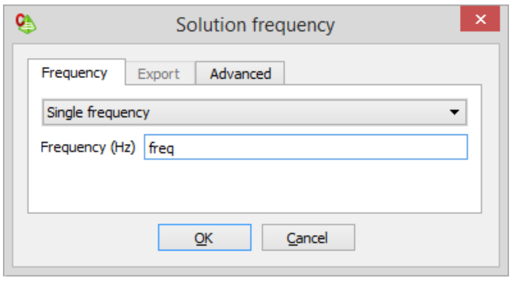


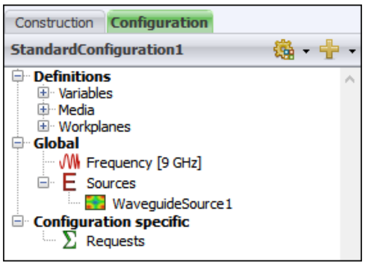
Sources may also be added to ports via the model tree. Right-click *Port1* in the model tree and select *Waveguide source* from the context menu.

Set the simulation frequency



Select the *Source/Load* tab and click the *Frequency* icon (*Settings* group) to open the *Solution frequency* dialog. A variable was created at the beginning of this example that contains the solution frequency. Enter *freq* in the *Frequency* field by selecting the editbox and then holding down <Ctrl><Shift> and clicking with the mouse the variable *freq* in the tree. With the frequency set to *freq*, the actual frequency is 9 GHz.



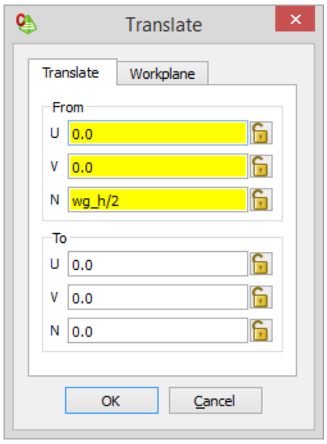


The solution frequency can be seen on the *Configuration* tab once the solution frequency dialog has been closed.

Define model symmetry

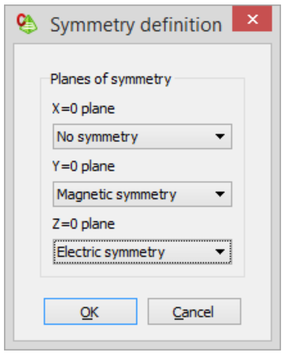
The model contains symmetry, but first the geometry must be moved so that symmetry can be utilised. The symmetry planes are only possible at X, Y or Z equal to zero.

Select the geometry (there should only be a single unioned part) in the model tree and click the *Transform* tab. Click the *Translate* icon (*Transform* group). Set the *From* and *To* fields as indicated.



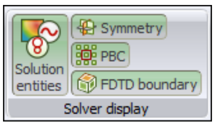
Group	*Field*	*Expression*
From	U	0
	V	0
	N	`wg_h/2`
To	U	0
	V	0
	N	0

This translation will move the structure down so that it is symmetrical around the Z=0 plane. Click the *OK* button to apply the translation and close the dialog.



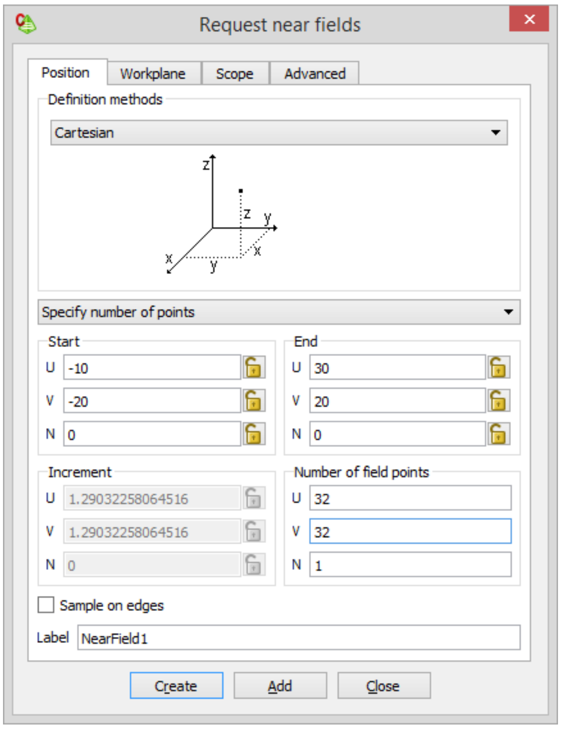
The model has magnetic symmetry at Y=0 and electric symmetry at Z=0. The required resources (memory and time) can be reduced by allowing the solution kernel to utilise symmetry. Note that this step is optional and the user is encouraged to remove symmetry later to compare the difference in solution time and memory usage.

Select the *Solve/Run* tab and click the *Symmetry* icon (*Solution settings* group). Set *Magnetic symmetry* at the Y=0 plane and *Electric symmetry* at the Z=0 plane.

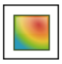


The symmetry display may be hidden by selecting the *Display options* tab in the *3D view* context. Click the *Symmetry* icon (*Solver display* group) to hide the symmetry in the 3D view.

Add a near field request



The near field values on a surface through the centre of the waveguide will now be calculated.



Select the *Request* tab and click the *Near fields* icon (*Solution requests* group) to display the *Request near fields* dialog. Enter the values as indicated.

Dimension	*Start*	*End*	*No. of Points*
U	-10	30	32
V	-20	20	32
N	0	0	1

Note that near field requests on PEC boundaries will result in a warning by the FEKO solution kernel, this is the reason why the option to *sample on edges* has been unselected.

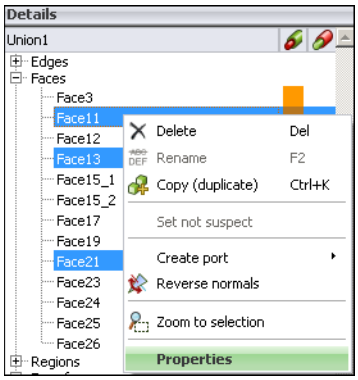
Leave the default label as *NearField1*.

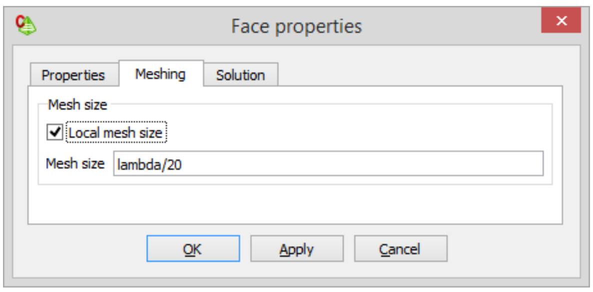
Click the *Create* button.

5.4 Mesh creation

The geometry was created, but still has to be meshed before the FEKO solver can be used to simulate the model. Three of the faces require a finer mesh than other faces. The faces that require a finer mesh are the ones that have waveguide ports on them (mesh should be fine enough to represent the highest mode that should be taken into account).

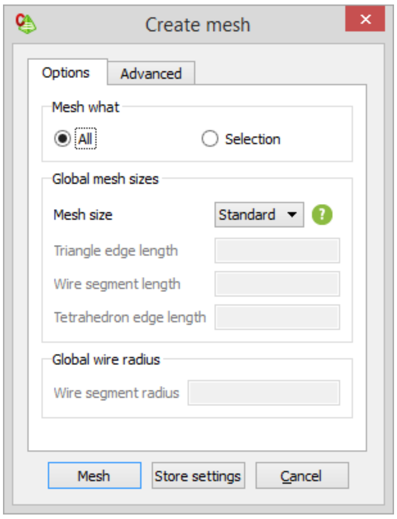
A finer mesh is required on the waveguide port faces. Select the geometry in the model tree. The three faces that must be refined can be selected in the details tree (or the 3D view). Right-click the faces (in the details tree) and select *Properties*.





Select the *Meshing* tab. Set the local mesh size to `lambda/20`. Click the *OK* button to close the dialog.

Now the model can be meshed. Select the *Mesh* tab and click the *Create mesh* icon (*Meshing* group). Use the default settings and click the *Mesh* icon to create a mesh and close the dialog. The shortcut key <Ctrl><M> may also be used to launch the *Create mesh* dialog.



Finally save the model as `gs_magicTwaveguide` (or any other file name) in a directory of your choice. The CADFEKO message windows will indicate that four files have been saved.

5.5 CEM validate

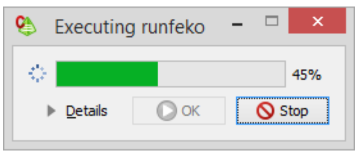
The model should now be computational electromagnetically validated. The purpose of this validation is to ensure that errors regarding the *Frequency*, *Geometry*, *Mesh* and *Solution* are found and corrected before the FEKO solver is run.

Run the *CEM validate* by selecting the *Solve/Run* tab and click the *CEM validate* icon (*Validate* group). When any errors are found, a respective error can be selected to display a short description of the error being shown in the *Error/Warning details of the selected item* window.

5.6 Obtaining a solution

After completing the model preparation, the solver should be invoked to calculate the requested results. The solver can be invoked in a number of ways, from a command window or from any one of the GUI components. In this example, the solver will be invoked from CADFEKO.

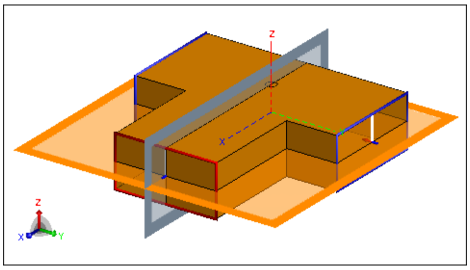
To invoke the solver from CADFEKO select the *Solve/Run* tab and click the *FEKO solver* icon (*Run/launch* group). A window will open, indicating the progress of the simulation.



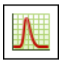
5.7 Visualisation of results

POSTFEKO is used to view and post-process all FEKO results. This user interface provides the viewing of 3D results (far fields, near fields, currents, etc.) with the geometry and the plotting and exporting of 2D graphic results.

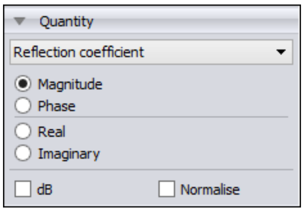
This example will show how to view the reflection coefficient at *Port1* on a 2D Cartesian graph and the near field values inside and outside the waveguide.



To launch POSTFEKO, click the *Run POSTFEKO* icon on the application launcher. POSTFEKO opens with the waveguide power divider displayed in the 3D view.



A new graph that plots information relating to the waveguide source is created by clicking the *Cartesian* icon (*Home* tab, *Create new display* group).



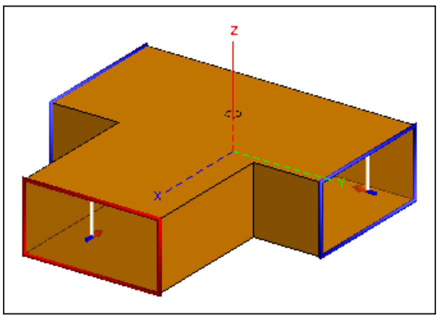
Add the reflection coefficient data point by clicking the *Source data* icon (*Home* tab, *Add results* group) and selecting *WaveguideSource1* from the dropdown menu. Reflection coefficient is often quoted in dB.

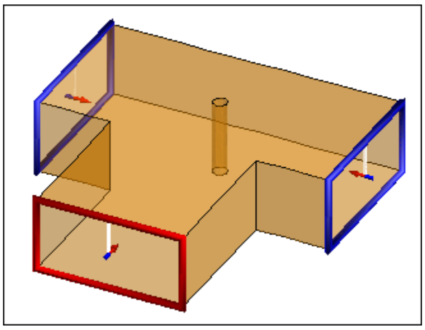
To set the vertical scale of the graph to dB, check the dB checkbox in the *Result palette*, *Quantity* section.

To add an annotation to the data point, select the *Measure* tab on the *Cartesian* contextual tab. Click the *Point* icon (*Custom annotations* group) and select *Global maximum* from the dropdown menu. The annotation shows that the power reflected back to the *Port1* is more than 30 dB lower than the power applied to the same port.

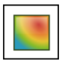
We now want to add the calculated near field values onto the 3D view. Select the 3D view by clicking the 3D view window tab.

The display of the symmetry planes may be disabled by selecting the *Display* tab on the *3D view* contextual tab. Click the *Symmetry* icon (*Method display*) to disable the display of the symmetry planes.





The mesh opacity of the waveguide may be enabled by clicking the *Mesh* tab of the *3D View* contextual tab. Click the *Mesh opacity* icon (*Opacity* group) and from the dropdown list, select a value (for example *40%*). If a specific value is required the value may be specified by selecting *Custom...* from the menu. The setting of the mesh opacity is required to be able to view the near field inside the waveguide.



We can now click the *Near field* icon (*Add results* group on the *Home* or *Result* tab). Select *NearField1* from the dropdown menu. The near field result is immediately displayed.

To animate the phase of the near field, in the *Result palette*, *Quantity* section, select *Instantaneous magnitude*. In the *3D View* context and select the *Animate* tab. Click the *Type* icon (*Settings* group). From the dropdown list select *Phase*. To start the animation process, click the *Play* icon (*Play* group). Click the *Play* icon again to terminate the animation.

5.8 Closing remarks

A waveguide section with three waveguide ports has been simulated. The time evolution of the near field values in a plane has been displayed with the 3D model.

Symmetry was utilised in two planes to reduce system requirements. Symmetry reduces both time and memory (RAM) required and is present in many real life problems. The user is encouraged to use symmetry when possible and also to investigate the difference in required resources when symmetry is enabled and disabled.

6 Getting started: Optimising the gain of a bent dipole

6.1 Example overview

The gain of a bent dipole placed in front of a square metallic reflector is maximised. The distance between the reflector and the bent dipole (d) as well as the dipole bend-angle (α) are varied in the optimisation. The goal is to maximise the maximum gain in the azimuth plane at a single frequency. The model geometry and related parameters are illustrated in Figure 6-1.

The model utilises symmetry to reduce memory requirements and to increase the calculation speed. Even though this is a small model that does not require a lot of resources, symmetry is used to illustrate the decreased resources required when utilising symmetry.

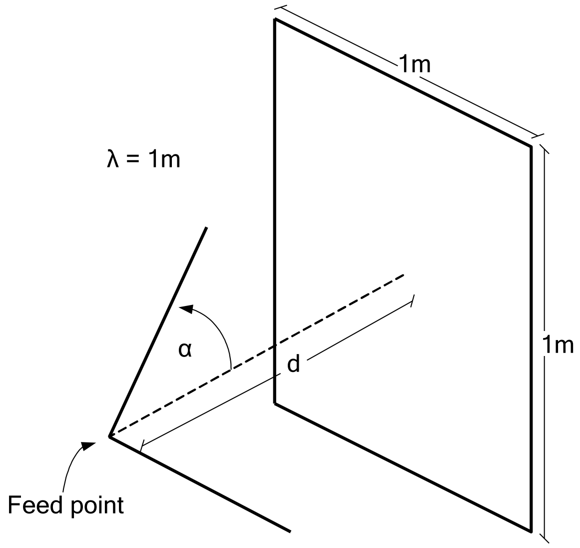


Figure 6-1: Sketch of the model.

6.2 Before starting the example

Before starting this example, please ensure that the system satisfies the minimum requirements before starting. A user should also ascertain whether the topics presented in this example are relevant to the intended application and FEKO experience level.

The topics demonstrated in this example are:

- Defining optimisation in CADFEKO.
- Running the FEKO optimiser (OPTFEKO).
- Adding a voltage source to a wire segment.
- Viewing optimisation results in POSTFEKO.

The requirements for this example are listed below.

- FEKO 2017.2 RELEASE or later should be installed[a] with a valid licence.
- It is recommended that the demo video be watched before attempting this example.
- Working through this example should not require more than about 30 minutes.

[a]See the FEKO Installation Guide to install Altair FEKO.

While working through this example, the steps should be followed sequentially, otherwise explanations may seem to be out of context.

The model referred to in this example can be found in the

```
examples/GetStarted_models/Project6-Optimisation_of_a_Dipole
```

directory of the Altair FEKO installation or downloaded from our website.

6.3 Creation of the model in CADFEKO

The first step in every FEKO solution is to construct the model. Start by launching CADFEKO which opens with the start page. Click *Create a new model* to create a new model. The model is created using CADFEKO and stored in the `*.cfx` file.

The steps that we are going to follow to create the model are:

- Define variables to parametrise the model.
- Add the reflector and bent dipole geometry.
- Add a source to the dipole.
- Set the frequency of the solution.
- Set the symmetry planes to use.
- Add a far field request.
- Add the optimisation request.

CADFEKO should now be open with a new, empty model.

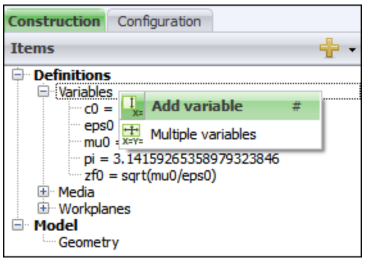
Define variables

Variables may be defined by selecting the *Construct* tab and clicking the *Variable* icon (*Define* group).

Variables may also be defined via the tree. Right-click the *Variables* group in the model tree and select *Add variable*. A short comment for a variable can be added to the *Comment* field. The inclusion of comments for variables is optional. The variable's comment is displayed when one hovers the mouse over the variable name (if available).

Create the following variables:

Name	*Expression*
alpha	60
alpha_rad	`alpha*pi/180`
d	0.25
lambda	1
freq	`c0/lambda`



Create reflector (rectangle)

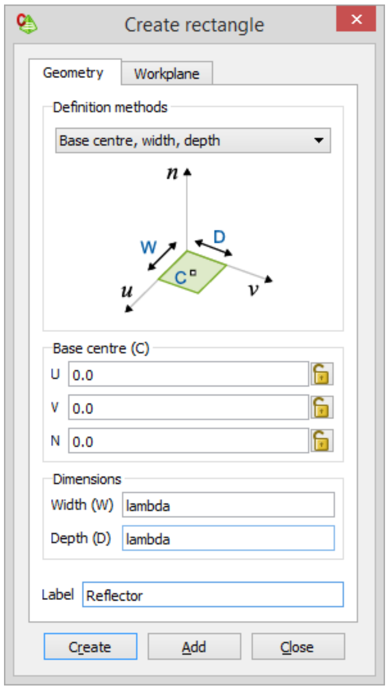
Create a rectangle by selecting the *Construct* tab and clicking the *Rectangle* icon (*Create surface* group). Select the *base centre, width, depth* definition method.

Enter the following values in the fields:

Field	*Value*
Width	`lambda`
Depth	`lambda`
Label	`Reflector`

The workplane of the *Reflector* will now be modified. Select the *Workplane* tab of the *Create rectangle* dialog. Below the *Rotate workplane* groupbox, click once the middle icon to rotate the workplane 90° around the V axis.

Click the *Create* button.

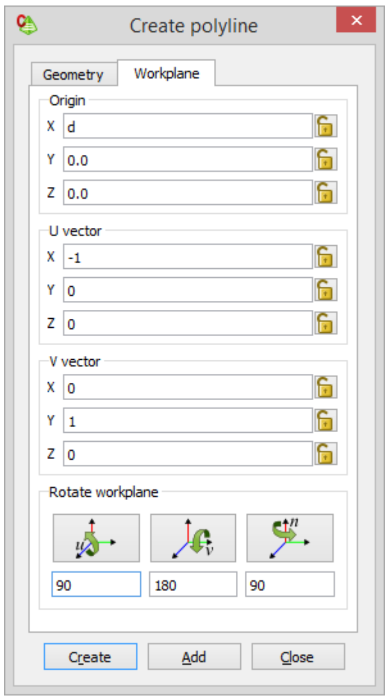


Create dipole (polyline)

Click the *Polyline* icon (*Create curve* group) to launch the *Create polyline* dialog.

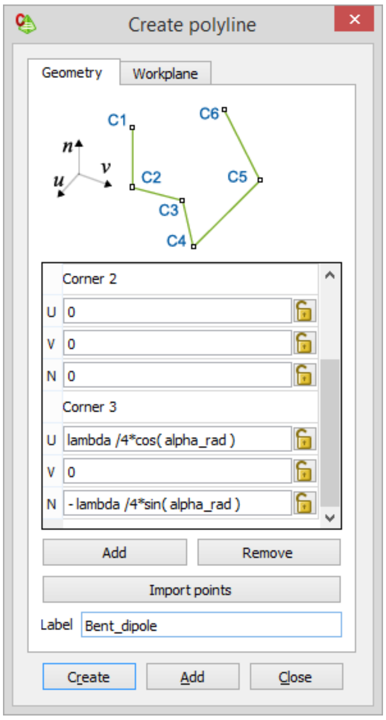
We are going to create the bent dipole on a new workplane so that it can be easily moved. Select the *Workplane* tab and set the local origin to (d, 0, 0) as indicated in the image. Note that variables and named points can also be entered by pressing <Ctrl><Shift> and clicking with the mouse on the variable or point in the tree.

Below the *Rotate workplane* groupbox, rotate the workplane by 180 degrees around the V axis by entering the angle of 180 and clicking the middle icon. The result of this action is shown in the image (right). Note the new orientation and position of the workplane in the 3D view. Return to the *Geometry* tab.

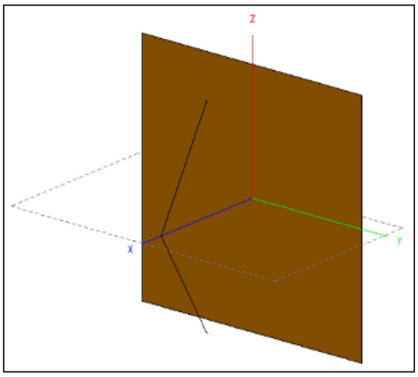


Create a polyline with the three corners listed in the table below. Use the *Add* button to add the third corner.

Corner	*Coordinate*	*Expression*
1	U	`lambda/4*cos(alpha_rad)`
	V	`0`
	N	`lambda/4*sin(alpha_rad)`
2	U	`0`
	V	`0`
	N	`0`
3	U	`lambda/4*cos(alpha_rad)`
	V	`0`
	N	`-lambda/4*sin(alpha_rad)`



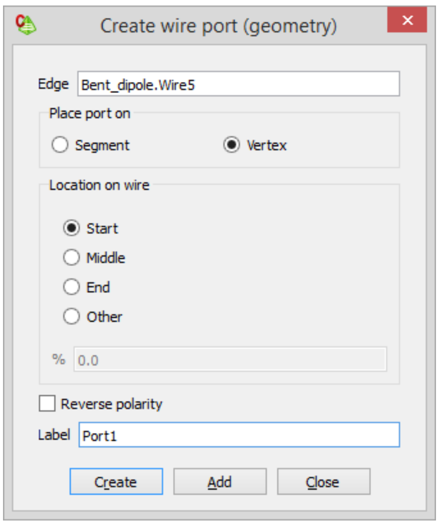
Name the polyline `Bent_dipole` by typing the name in the *Label* field. Click *Create* to create the polyline and close the dialog.

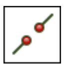


If all the steps have been followed correctly, the created geometry should be similar to this image.

Define the port

We have now created the geometry and need to add a port where the dipole can be excited.



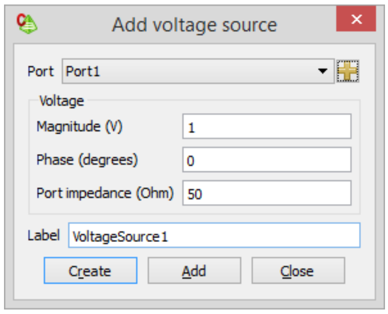


Select the *Source/Load* tab and click *Wire port* (*Ports* group).

The *Create wire port* dialog is displayed. Ensure that the *Edge* field is yellow to indicate that the field can be populated by selecting an item with the mouse. Use the mouse to select the top part of the *Bent_dipole* in the 3D view.

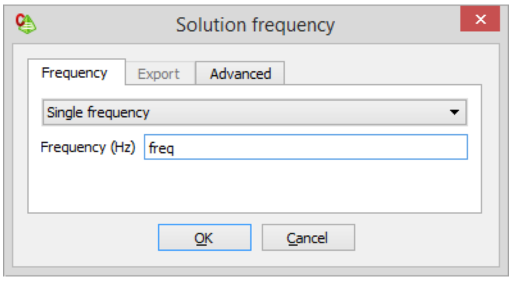
Set the port location to the *Start* of the *Vertex*. A completed *Create wire port* dialog is shown in the image, please ensure that all settings are correct. It should be noted that the name of the wire edge may differ, but it should be *Bent_dipole.Wire* followed by a number.

Click *Create* to add the port and close the dialog. This should result in a port at the centre of the dipole on the 3D window.

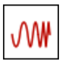


Add the voltage source

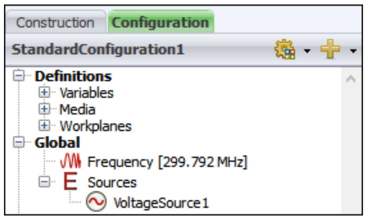
Add a voltage source to the port by selecting the *Source/Load* tab and click the *Voltage source* icon (*Sources on ports* group). Create a voltage source on *Port1*.



Define the simulation frequency

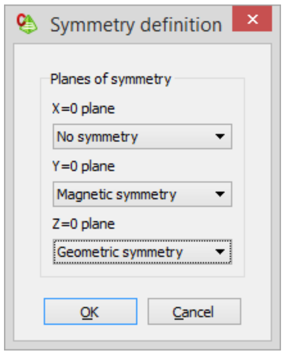


Click the *Frequency* icon to open the *Solution frequency* dialog. A variable was created at the beginning of this example that contains the solution frequency. Enter *freq* in the *Frequency* field.



With the frequency set to *freq*, the actual frequency is 299.792 MHz. This can be seen in the tree once the solution frequency dialog has been closed.

Define model symmetry

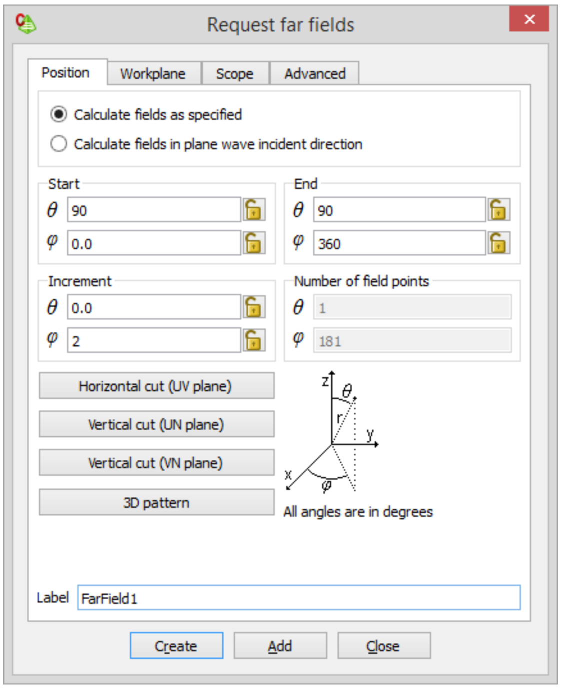


The model has magnetic symmetry at Y=0 and geometric symmetry at Z=0. The required resources (memory and solution time) can be reduced by allowing the solution kernel to utilise symmetry. Note that this step is optional and the user is encouraged to remove symmetry later and to compare the difference in solution time and memory usage.

Select the *Solve/Run* tab and click *Symmetry*. Set the *Magnetic* symmetry at the Y=0 plane and *Geometric symmetry* at the Z=0 plane. Click the *OK* button to define the symmetry and close the dialog.

Setup the solution request

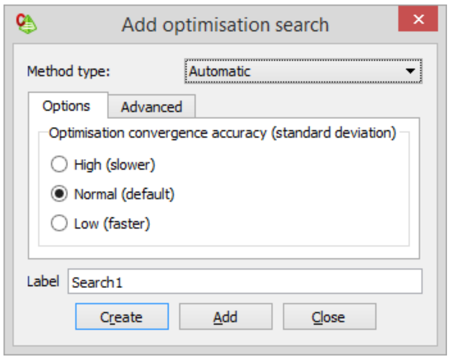
The far field gain in the azimuth direction is going to be optimised. This means that a far field request in the azimuth direction has to be requested.

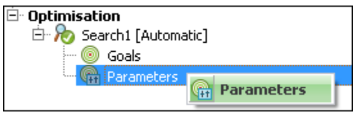


Select the *Request* tab and click the *Far fields* icon (*Solution requests* group). Click the *Horizontal cut (UV plane)* to request the far field gain to be calculated in the azimuth direction. Leave the default label as *FarField1*. Click *Create* to create the far field request and close the dialog.

Define the optimisation search

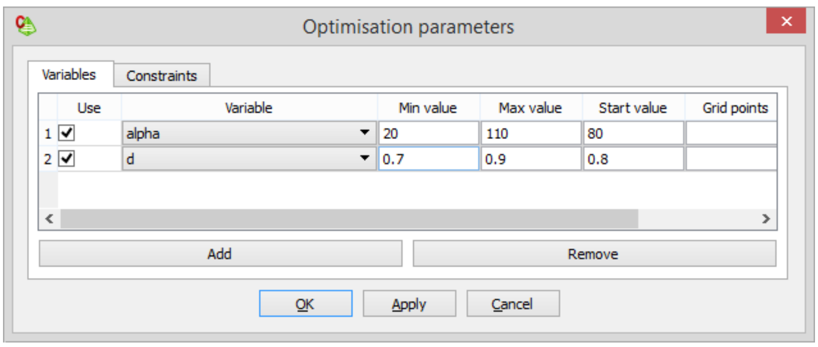
Select the *Request* tab and click the *Add search* button (*Optimisation* group) to launch the *Add optimisation search* dialog. Set the optimisation convergence accuracy to *Normal (default)*. The other settings should not be changed. We are using the *Automatic* setting as method type. This allows FEKO to decide what method to use. Click *Create* to create a new optimisation search.





Right-click *Parameters* in the model tree. Click *Parameters* in the context menu.

The *Optimisation parameters* dialog is displayed. Use the *Add* button to add another parameter. The two parameters that should be selected is *d* and *alpha*. Setup the optimisation parameters as indicated in the image. Click the *OK* button once the setup is complete.



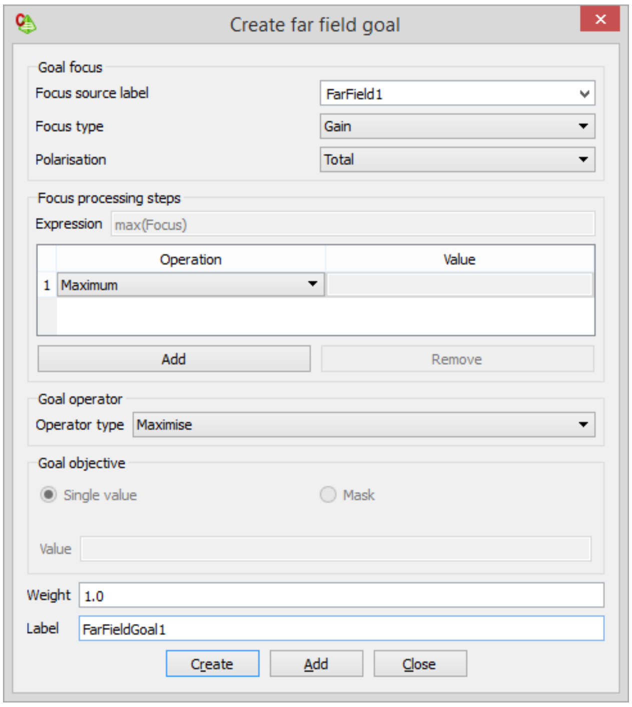
We want to maximise the far field gain in the azimuth plane. We have already requested that the far field should be calculated. We now set up the optimisation goal.

Ensure that the optimisation search, *Search1*, is selected in the tree and then select the *Request* tab and click the *Add goal function* icon (*Optimisation* group). From the dropdown menu select *Far field goal*. Note that the goals will be disabled if the optimisation is not currently selected in the tree.

The *Create far field goal* dialog is divided into a few sections that will briefly be discussed.

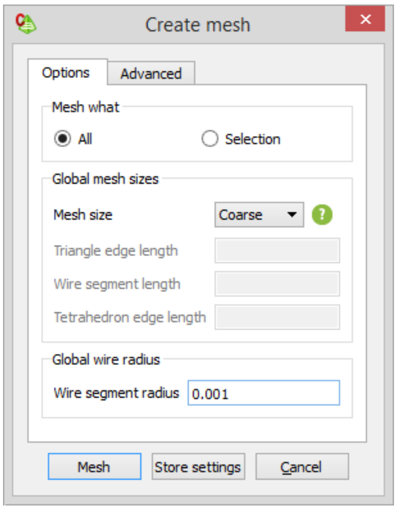
- The first section is the *Goal focus* and is the calculation that should be performed by the FEKO solver. In our case, we select *FarField1* that was created earlier. We also select the component that we are interested in, namely the *Gain* of the far field. Select *Total* as the polarisation type.
- The *Focus processing steps* allow processing of the focus before it is compared to the *objective*. Select *Maximum* to use only the maximum value of the total far field gain.
- The *Goal operator* describes how the objective and focus is compared. We want to maximise the gain, so select *Maximise* as a goal operator.
- *Goal objective* is what the value of the focus is compared to. Note that there is no *Goal objective* for maximisation or minimisation.
- A *Weight* can be defined for each goal. This weighting is used to modify the contribution of the goal's error to the global error during the fitness evaluation.

Enter all the fields as they are in the image.

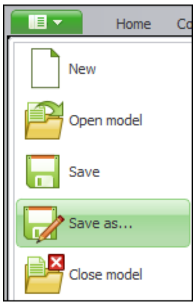


6.4 Mesh creation

The geometry has been created, but still has to be meshed before the FEKO solver can be run.



Select the *Mesh* tab and click the *Create mesh* icon (*Meshing* group) or use the shortcut key <Ctrl><M>. For the *Mesh size* set the size as *Coarse*. Set the *Wire segment radius* as 0.001 and click the *Mesh* icon.



Finally save the model as `Dipole_Optimisation` (or any other file name) in a directory of your choice. Click the *Application menu* and from the dropdown menu click the *Save as* icon. The CADFEKO message windows will indicate that six files have been saved.

6.5 Obtaining a solution and displaying the results

The model has now been completely configured and is ready to be solved and viewed. The steps required to view the results are:

- Run the CEM validate.
- Run the FEKO solver once to test the model and produce initial output for POSTFEKO.
- Open POSTFEKO and configure the views that we want to look at during optimisation.
- Launch the optimisation.
- View the optimisation results.

The model should now be computational electromagnetically validated. The purpose of this validation is to ensure that errors regarding the *Frequency*, *Geometry*, *Mesh* and *Solution* are found and corrected before the FEKO solver is run.

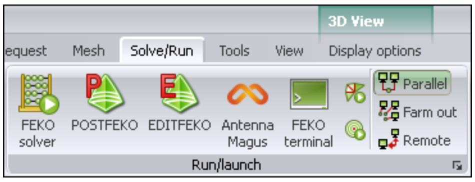
CEM validation

Select the *Solve/Run* tab and click the *CEM validate* icon (*Validate* group).

When any errors are found, an error can be selected, resulting in a short description of the error being shown in the *Error/Warning details of the selected item* window.

Run FEKO solver (not the optimisation)

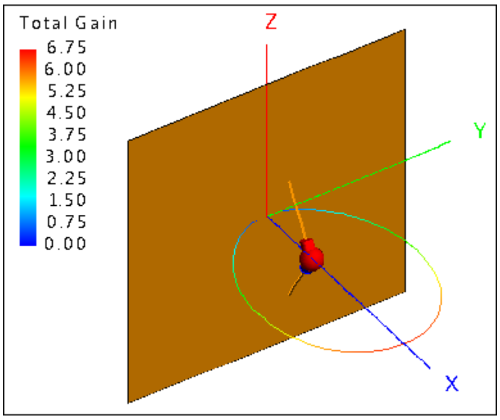
Next, a single FEKO run is performed. The shortcut keys <Alt><4>, the application launcher or the ribbon can be used to launch the FEKO solver. Select the *Solve/Run* tab and click the *FEKO solver* icon (*Run/Launch* group). This should not take longer than a few seconds and no warnings or errors should be reported. Close the message window once the run has completed.

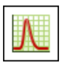


Set up graphs and views in POSTFEKO

Run POSTFEKO by using the <Alt><3> shortcut key, application launchers or from the CADFEKO ribbon. POSTFEKO opens by default with a single 3D window that displays the geometry of the model.

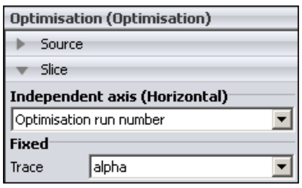
Click the *Far field* icon (*Home* tab, *Add results* group) and from the dropdown menu select *FarField1*.





Add a 2D optimisation graph by selecting the *Home* tab and clicking the *Cartesian* icon (*Create new display* group).

Add the optimisation result by clicking the *Optimisation* icon (*Add results* group) and selecting *Optimisation* from the dropdown list.



In the *Result palette*, *Slice* section, set the *Trace* to *alpha*.

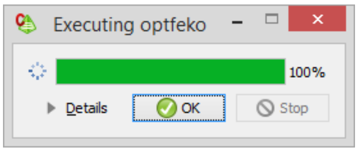
Create two additional Cartesian graphs and in the *Result palette* set the *Trace* to *d* and *search1.goals.farfieldgoal1* respectively.

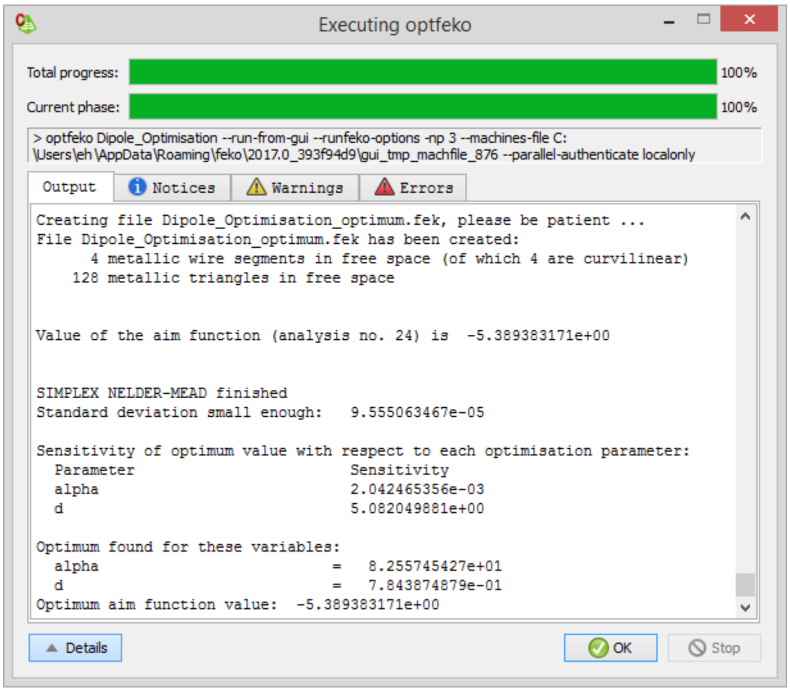
The four windows can be neatly positioned for easy viewing during optimisation by selecting the *Tile* icon (*View* tab, *Window* group).

Run the optimisation

All that remains is to run OPTFEKO. On the *Home* tab, click the *OPTFEKO* icon (*Run/launch* group) or use the application launcher. POSTFEKO will automatically update the displayed graphs.

The run dialog is displayed in a condensed format. It is advisable to view any warnings, errors or notes that are indicated in the dialog by clicking the *Details* button. The convergence information as well as the parameters of the optimal solution is displayed in the run dialog at the end of the optimisation run. Click the *Details* button to see the complete list of output.

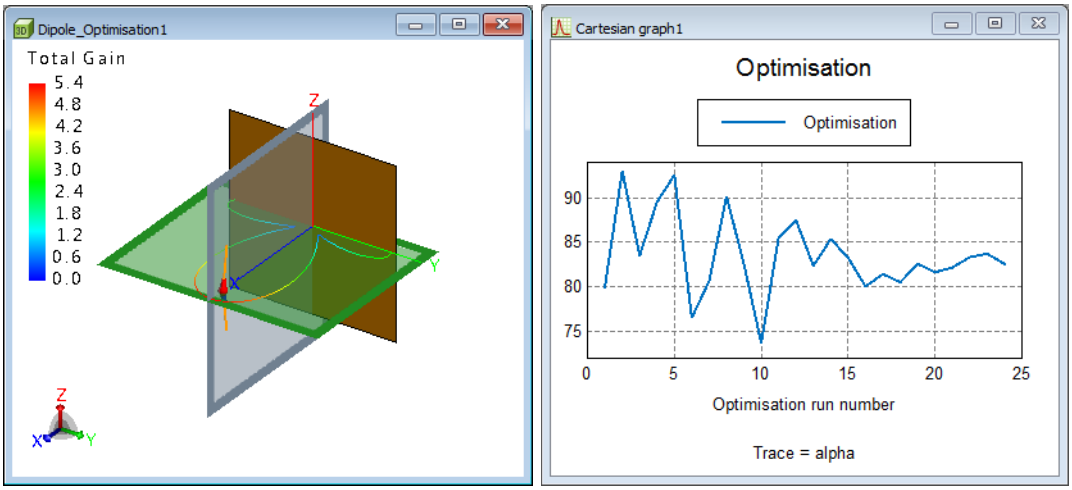




After the optimisation has completed, the message window displays the optimisation results. We see that the optimum value for ‘alpha’ is 82.557° and 0.784 m for ‘d’. With these values, the maximum gain is 5.41 (linear). When the optimiser has enough information to do a sensitivity analysis, these results will also be displayed.

View optimisation results in POSTFEKO

In POSTFEKO all the graphs have now been populated with the details of the optimisation process. We can clearly see how the goal that we setup has reached a maximum value at the end of the optimisation. For more details on the optimisation process, a log file named `gs_dipole.log` has been created in the project directory.



The optimiser creates a new CADFEKO model (`*.cfx`) with the variables set to the optimum values. This model is located in the same directory as the current model, but the name has been extended with `_optimum`.

6.6 Closing remarks

This example has demonstrated how to configure a CADFEKO model as well as how optimisation in CADFEKO is used. The optimisation process as well as the optimum values for the model parameters were displayed in POSTFEKO, but can also be viewed in the log file.

Index