RF in DxDesigner® - Expedition® Flow User's Manual

Software Version EE 7.9.5

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/
Send Feedback on Documentation: supportnet.mentor.com/doc_feedback_form

Table of Contents

List of Figures

List of Tables

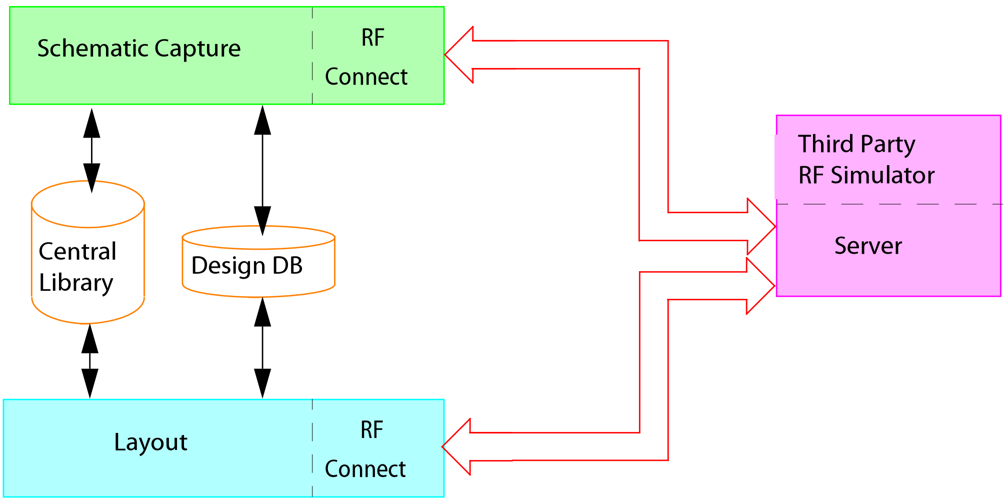
Chapter 1
RF Flow Introduction

RF Design Process Overview

The Radio Frequency (RF) tool in the Expedition Enterprise flow, enables you to create RF designs. You can also simulate the RF design with a third party RF simulator by transferring design data over a bidirectional dynamic link.

Using these RF tools, you can edit and simulate your designs, while maintaining concurrency in schematic capture and layout. The transfer of design data over a dynamic link provides a secure, real-time integration, and a seamless process between schematic capture, layout, and the simulator (Figure 1-1).

Figure 1-1. EE RF Flow



The RF design flow supports the following third party RF simulators:

- Agilent Technologies' Advanced Design System (ADS) RF Simulator (ADS RF Simulator).
- Applied Wave Research (AWR) Microwave Office (MWO) RF Simulator (MWO RF Simulator).

RF tools enable you to:

- Transfer data bidirectionally (Connecting Schematic Capture to RF Simulator Server and Connecting Layout to RF Simulator Server).

- Group RF shapes and elements in the schematic (Creating an RF Group) and send the RF group to the RF simulator via the dynamic link. You can make design changes in the RF simulator and post back the changes to the schematic capture tool (Post Back to Schematic Capture and Layout and Posting Back MWO Updates to Layout).
- Use RF shapes in the Central Library to create a design in the RF simulator (Placing RF Symbols). You can send the RF simulator design to schematic capture where it becomes part of an existing design.
- Forward Annotate (FA) the schematic capture design to layout. Use the RF Toolkit in layout to add and create meanders (traces), vias, and RF shapes (RF Layout). You can send an RF group to the RF simulator and make design changes (Sending Layout Data). Post back the RF simulator changes to incorporate the changes into the layout design ().

DxDesigner RF Introduction

You must acquire a DxRFEngineer license to access the RF tools (Acquiring an RF License). You can place RF shape symbols in the same manner as any other symbol (Placing RF Symbols).

Develop an RF design in the schematic capture tool as follows:

- Place RF simulator RF shapes in the schematic design (Adding ADS RF Shapes to Central Library and Adding AWR/MWO RF Shapes to Central Library).
- Check the RF designs (RF DRC).
- Edit the RF shape parameters (RF Parameters Dialog Box - Schematic Capture).
- Set the following RF design defaults:

  - Parameter Units (Setting the RF Parameters Default Units of Measure)
  - Min/Max Frequency (Setting RF Frequency Range)
  - Substrate (Adding a Substrate)
- Create variable blocks, which enables use of parameter variables for RF shapes. (Creating Variables in RF Shape Parameters).
- Transfer design data between the schematic capture tool and the RF simulator with the dynamic link. (Connecting Schematic Capture to RF Simulator Server).

Related Topics

Getting Started

Expedition RF Introduction

You must have an Advanced Technology Pro (RF) license to access the RF Toolkit (Advanced Technology Pro (RF)). Use the RF tools to create a mixed technology design that includes RF, digital, and analog (RF Toolkit Toolbar). The RF Toolkit provides access to the layout RF tools.

Use the following to develop an RF design in the layout tool:

- Create variable blocks to define RF shape parameters (Adding a Varblock).
- Place RF shapes with the auto arranger (Placing RF Groups with Auto Arranger).
- Create meanders (traces) (Meander Properties Dialog Box).
- Place RF stitched vias (Vias in RF Designs).
- Set RF shape clearance rules in X, Y, and Z axes (Clearance Editor Dialog Box).
- Define RF shape entry angles (RF Entry Rules Dialog Box).
- Change RF shape parameters (RF Shape Parameters Dialog Box).
- Transfer design data between the layout tool and the RF simulator with the dynamic link (Connecting Layout to RF Simulator Server).

Related Topics

Getting Started

Manufacturing Data

RF circuits require special attention to the precision of the output data. Typically, the dimensions are much smaller than analog and digital designs and require a more accurate format (Generating Manufacturing Data).

Related Topics

Getting Started

Getting Started

The following is an overview of the RF design process. Clicking a link takes you to detailed information on the selected subject matter.

- Acquiring an RF licence:
    - DxDesigner — Acquiring an RF License
    - Expedition — Opening a PCB for RF Design
- Supported RF simulators:
    - Agilent Technologies' Advanced Design System (ADS) RF Simulator — ADS RF Simulator
    - Applied Wave Research (AWR) Microwave Office (MWO) RF Simulator — MWO RF Simulator
- Installing RF simulator elements into the central library:
    - Adding ADS RF Shapes to Central Library
    - Adding AWR/MWO RF Shapes to Central Library
- Grouping the design by RF functions or regions:
    - DxDesigner — RF Group/Ungroup Dialog Box
    - Expedition — Group/Ungroup Dialog Box
- Setting up the bidirectional dynamic link:
    - The RF simulator provides a server:
        - ADS RF simulator — Launching the ADS Server
        - AWR RF Simulator — Launching MWO Server
    - DxDesigner and Expedition connecting to RF simulator server:
        - DxDesigner — Connecting Schematic Capture to RF Simulator Server
        - Expedition — Connecting Layout to RF Simulator Server
- Sending the RF design group to the RF simulator:
    - DxDesigner — RF Group/Ungroup Dialog Box Treeview Popup Menu Items
    - Expedition — Sending Netlist - Layout
- Sending the RF simulator data back to DxDesigner or Expedition for incorporation:
    - DxDesigner
        - Sending ADS Design to Schematic Capture Tool

    - Sending an MWO Design to Schematic Capture
- Expedition
    - Post Back to Schematic Capture and Layout
    - Posting Back MWO Updates to Layout

In addition to this document, a the following manual provides a description of the ADS third party RF simulator elements:

- *RF Design Tools User's and Reference Manual Supplement: Supported ADS Library Elements*

Related Topics

RF Design Process Overview

Chapter 2
Schematic Capture

Use the schematic capture tool to place RF symbols in a design when you have an RF license.

The following topics describe how to create an RF design:

- Acquiring an RF License
- Setting the RF Parameters Default Units of Measure
- Setting RF Frequency Range
- Adding a Substrate
- Editing a Substrate
- Placing RF Symbols
- Creating an RF Group
- Moving Members Between RF Groups
- Adding an RF Varblocks
- Creating Variables in RF Shape Parameters
- Changing RF Symbol Parameters
- RF DRC
- Sending Netlist
- Sending Schematic Data

Acquiring an RF License

You must have an RF license to access the RF toolbar and menus in the schematic capture tool.

Prerequisites

- The schematic capture tool must be open.

Procedure

1. Choose **Setup** > **Settings**.
2. Select Licensing category.
3. Choose DxRFEngineer.
4. Click **OK**.

Results

The RF Toolbar appears in the schematic capture tool (RF Toolbar).

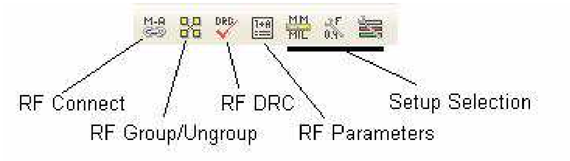
Related Topics

Setting the RF Parameters Default Units of Measure
Setting RF Frequency Range
Adding a Substrate
Editing a Substrate

RF Toolbar

Access RF functions through the RF toolbar. Display the RF toolbar by selecting **View > Toolbars > RF** (Figure 2-1).

Figure 2-1. RF Toolbar



The RF Toolbar provides access to the following:

RF Connect — RF Connect Functionality

RF Group/Ungroup — Creating an RF Group

RF DRC — RF DRC

RF Parameters — Changing RF Symbol Parameters

Setup Selection — RF Toolbar Setup Selections

Related Topics

Acquiring an RF License

RF Toolbar Setup Selections

Use the setup selections to define units of measure, frequency range, and substrates. Typically, you define these items before you place RF symbols.

Note

Changes to setup selections are not retroactive. Therefore, it is strongly recommended that you do not change any setup selections values after placing RF symbols.

The RF Toolbar setup selections provide the following:

Default Units () — Setting the RF Parameters Default Units of Measure

Frequency Range () — Setting RF Frequency Range

Substrates () — Adding a Substrate and Editing a Substrate

Related Topics

RF Toolbar

Acquiring an RF License

Placing RF Symbols

Default Units

To access:

- Click **Default Units** ( ) (RF Toolbar)
- Choose **View** > **RF** > **Default Units**

Description

Use the Setup Default Units dialog box to define the unit of measure for RF symbol parameters (Figure 2-2). To define a unit of measure, refer to Setting the RF Parameters Default Units of Measure.

Figure 2-2. Setup Default Units Dialog Box

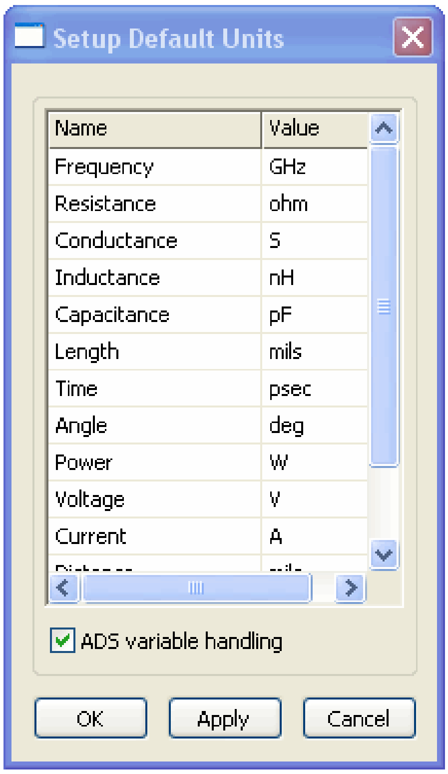


Table 2-1. Setup Default Units Dialog Box Content

Field	Description
Name	Displays the RF parameter name (refer to Table 2-2).
Value	Defines the default unit value (refer to Table 2-2).

Table 2-1. Setup Default Units Dialog Box Content (cont.)

Field	Description
ADS variable handling	Defines the variable evaluation method and checks compatibility with Advanced Design System (ADS) (ADS RF Simulator). Checked, converts the base unit value for the evaluation result to the default value. Unchecked, prevents the conversion of the base unit to the default value in an expression. **Note**: Refer to the following for ADS variable handling examples: • Calculation Example with Defined Unit • Calculation Example with Undefined Unit

Table 2-2. Setup Default Units Dialog Box - Supported Parameter Values

Parameter Name	Base Unit	Default Value	Supported Values
Frequency	Hertz	GHz	GHz, Hz, MHz, PHz, THz, cHz, fHz, mHz, nHz, pHz, uHz
Resistance	Ohm	Ohm	Gohm, Kohm, Mohm, Pohm, Tohm, cohm, fohm, mohm, nohm, ohm, pohm, uohm
Conductance	Siemen	S	GS, KS, MS, PS, S, TS, cS, fS, mS, nS, pS, uS
Inductance	Henry	nH	GH, H, KH, MH, PH, TH, cH, fH, mH, nH, pH, uH
Capacitance	Farad	pF	F, GF, KF, MF, PF, TF, cF, fF, mF, nF, pF, uF
Length	Meter	mil	Gm, Km, Mm, Pm, Tm, cm, fm, ft, in, m, mi, mils, mm, nm, pm, um
Time	Second	psec	Gsec, Ksec, Msec, Psec, Tsec, csec, fsec, hr, min, msec, nsec, psec, sec, usec
Angle	Radian	Deg	deg, rad
Power	Watt	W	GW, KW, MW, PW, TW, W, cW, dBm, fW, mW, nW, pW, uW
Voltage	Volt	V	GV, KV, MV, PV, TV, V, cV, fV, mV, nV, pV, uV
Current	Ampere	A	A, GA, KA, MA, PA, TA, cA, fA, mA, nA, pA, uA
Distance	Meter	mil	Gm, Km, Mm, Pm, Tm, cm, fm, m, mm, nm, pm, um
Temperature	Celsius	C	C
Attenuation	Decibel	dB	dB

Related Topics

Setting RF Frequency Range
Adding a Substrate
Editing a Substrate
Placing RF Symbols
RF Toolbar Setup Selections

Calculation Example with Defined Unit

This example demonstrates the variable evaluation method with a defined unit of measure (Parameter = Length; Value = mil). Results are shown with **ADS variable handling** checked and unchecked.

Description
Variable V1 and component C1 belong to the same group.

Variable V1
A=100 mil

Component C1
L=A

Default unit
Length (1 mil = 2.54e-5 meter)

ADS variable handling checked

1. The variable evaluation equation is: 100 * 2.54e-5.
2. With the length (L) as a multiplier in the expression, all units are converted to the base unit for calculations.
3. Result: 0.002540 meters
4. Convert the result from the base unit (meter) to the default value (mil).
5. Result: 100 mil

ADS variable handling unchecked

1. The variable evaluation equation is: 100 * 2.54e-5.
2. With the length (L) as a multiplier in the expression, all units are converted to the base unit for calculations.
3. Result: 0.002540
4. The result does not contain a unit of measure, which prevents conversion to the default value.

Related Topics

Calculation Example with Undefined Unit

Setting the RF Parameters Default Units of Measure

Setting RF Frequency Range

Adding a Substrate

Editing a Substrate

Placing RF Symbols

Calculation Example with Undefined Unit

The following example demonstrates the variable evaluation method with **no** defined unit of measure. Results are shown with **ADS variable handling** checked and unchecked.

Description
Variable V1 and component C1 belong to the same group.

Variable V1
A=100+100

Component C1
L=A

Default unit
Length

ADS variable handling checked

1. The plus (+) operator indicates that the value of the base unit (A) is 200.
2. Convert the result from the base unit (meter) to the default value (mil) (1 meter = 39370.0787 mil).
3. Result: 7874015.74 mil.

ADS variable handling unchecked

1. The plus (+) operator calculates the value to 200.
2. Result: 200.
3. The result does not contain a unit of measure, which prevents conversion to the default value.

Related Topics

Calculation Example with Defined Unit
Setting the RF Parameters Default Units of Measure
Setting RF Frequency Range
Adding a Substrate
Editing a Substrate
Placing RF Symbols

Substrates

To access:

- Click **Substrates** ( ) (RF Toolbar)
- Choose **View** > **RF** > **Substrates**

Description

Use the Setup Substrates dialog box to define the default substrate for RF symbols.

Table 2-4 lists the default substrates.

Figure 2-3. Setup Substrates Dialog Box

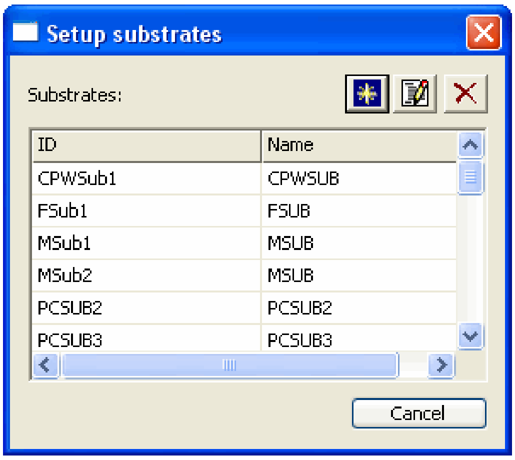


Table 2-3. Setup Substrates Dialog Box Content

Selection	Description
New	Creates a new substrate from a selected substrate or from the default substrate if none is selected (Adding a Substrate).
Edit ( )	Enables editing for the selected substrate (Editing a Substrate).
Delete	Deletes the selected substrate. Default substrates cannot be deleted (Table 2-4).
ID	Specifies an ID for the substrate. You can assign multiple IDs to the same substrate name.
Name	Specifies the substrate name.

Table 2-4. Alphabetical List of Default Substrates

CPWSUB

Table 2-4. Alphabetical List of Default Substrates (cont.)

FSUB
MLSUBSTRATE2
MLSUBSTRATE3
MLSUBSTRATE4
MLSUBSTRATE5
MLSUBSTRATE6
MLSUBSTRATE7
MLSUBSTRATE8
MLSUBSTRATE9
MLSUBSTRATE10
MLSUBSTRATE12
MLSUBSTRATE14
MLSUBSTRATE16
MSUB
PCSUB1
PCSUB2
PCSUB3
PCSUB4
PCSUB5
PCSUB6
PCSUB7
SSSUB
SSUB
SSUBO

Related Topics

Setting RF Frequency Range

Setting the RF Parameters Default Units of Measure

Placing RF Symbols

RF Toolbar Setup Selections

Setting the RF Parameters Default Units of Measure

You can set the default unit of measure for any RF symbol parameter.

Use the following procedure to set the default unit for an RF symbol parameter. The following example uses the Frequency parameter value.

Prerequisites

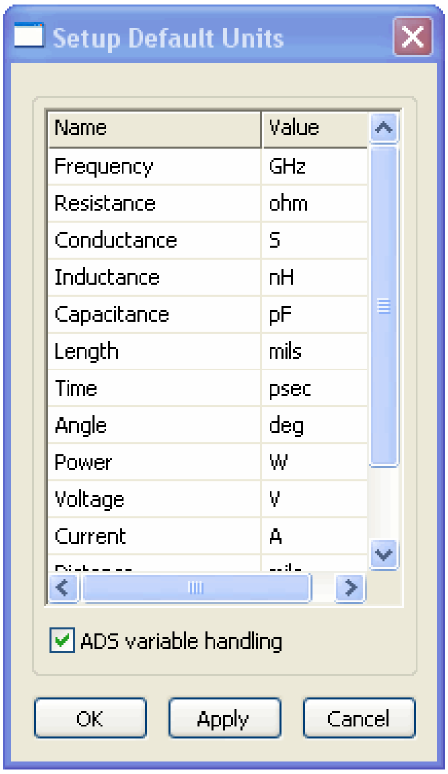
- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

1. Click **Default Units** () in the RF toolbar.
2. Click the Value cell for the Frequency parameter.
3. Select the new unit of measure from the dropdown list (Figure 2-4).

Tip: Stretch the dialog box horizontally so you can see both columns as you make changes.

Figure 2-4. Setup Default Units Dialog Box - Frequency



4. Click **Apply** to save and continue making changes, or click **OK** to save and exit.

Results

The default unit of measure for Frequency changes to the selected value.

Related Topics

Setting RF Frequency Range

Adding a Substrate

Editing a Substrate

Placing RF Symbols

Default Units

RF Toolbar Setup Selections

Setting RF Frequency Range

Use this procedure to define the frequency range of the RF design.

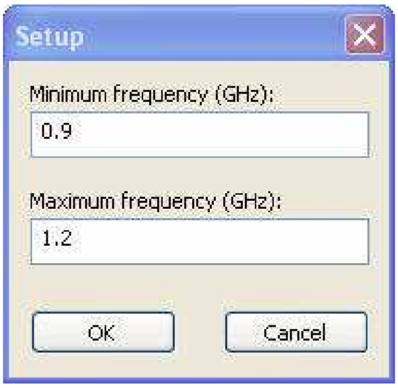
Prerequisites

- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

1. Click **Frequency Range** ( ) in the RF toolbar (Figure 2-5).

Figure 2-5. Setup Dialog Box



2. Enter the minimum and maximum values (in GHz) for the frequency range.

Note

The values are in GHz. You can change the unit of measure for Frequency in the Setup Default Units dialog box.

3. Click **OK**.

Results

The frequency range of the RF design is defined.

Related Topics

Adding a Substrate

Setting the RF Parameters Default Units of Measure

RF Toolbar Setup Selections

Editing a Substrate

Placing RF Symbols

Adding a Substrate

Substrates determine RF symbol characteristics. The RF tool set includes a group of predefined substrates (Table 2-4). You can also create a custom substrate using one of the predefined substrates as a seed.

Prerequisites

- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

1. Click **Substrates** ( ) in the RF toolbar.
2. Select an existing substrate that closely matches the parameters you want in your new substrate
3. Click **New** ( ).
4. Rename the ID and Name of the RF substrate.
5. Make additional changes to the Values in the Select parameters table to define the new substrate.
6. Click **Apply**.

Results

The new substrate appears in the Substrates list after the substrate you originally selected as the seed.

Note
You cannot reorder the list of substrates.

Related Topics

Editing a Substrate
Setting the RF Parameters Default Units of Measure
Substrates
Setting RF Frequency Range
Placing RF Symbols
RF Toolbar Setup Selections

Editing a Substrate

Use the following procedure to change an RF substrate's parameters.

Prerequisites

- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

1. Click **Substrates** ( ) in the RF toolbar.
2. Select the substrate you wand to edit in the Substrates table.
3. Click **Edit** ( ). The Substrates properties dialog box displays.
4. Click in the Value cell for the parameter you want to change.
5. Under the Parameters section, enter a new value in the Value text box.
6. Select the appropriate unit of measure in the Units dropdown list.
7. Click **Apply**.

Results

The changes are saved.

Related Topics

Adding a Substrate
Setting the RF Parameters Default Units of Measure
RF Toolbar Setup Selections
Setting RF Frequency Range
Placing RF Symbols

Placing RF Symbols

Create RF circuits in the schematic capture tool by placing RF symbols in your design.

Use the following procedure to place RF symbols in your design.

Prerequisites

- A schematic capture project must be open.
- RF symbols must be in the Central Library (Adding ADS RF Shapes to Central Library and Adding AWR/MWO RF Shapes to Central Library).

Procedure

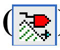
1. Click **DxDataBook (****)** in the DxDesigner toolbar.
2. Select the **CL View** tab.
3. Choose the **Symbol View** tab (Figure 2-6).

Figure 2-6. DxDataBook - Symbol View Window

4. Expand the RF partition that contains the RF symbol you want to place.
5. Select the RF symbol from the expanded list.
6. Place the RF symbol by dragging the symbol from the Place Symbol window into the drawing workspace and dropping it at the correct location.

Results

The RF symbol appears in the schematic.

Related Topics

Creating an RF Group

Creating Variables in RF Shape Parameters

Adding an RF Varblocks

RF DRC

RF Group/Ungroup Dialog Box

To access:

- Click **RF Group/Ungroup** () (RF Toolbar)
- Choose **View > RF > RF Group/Ungroup**.

Description

Use the RF Group/Ungroup dialog box to group or ungroup RF and non-RF symbols. You can group relevant RF circuitry by functionality or area. A symbol can only belong to one RF group, but a group can span several sheets and multiple schematics. You can create both RF groups and RF subgroups within RF groups. You can create any number of RF subgroups (Figure 2-7).

Note

Adding non-RF symbols into an RF group does not convert them to RF symbols. They remain regular symbols that are members of an RF group or subgroup.

Figure 2-7. RF Group/Ungroup Dialog Box

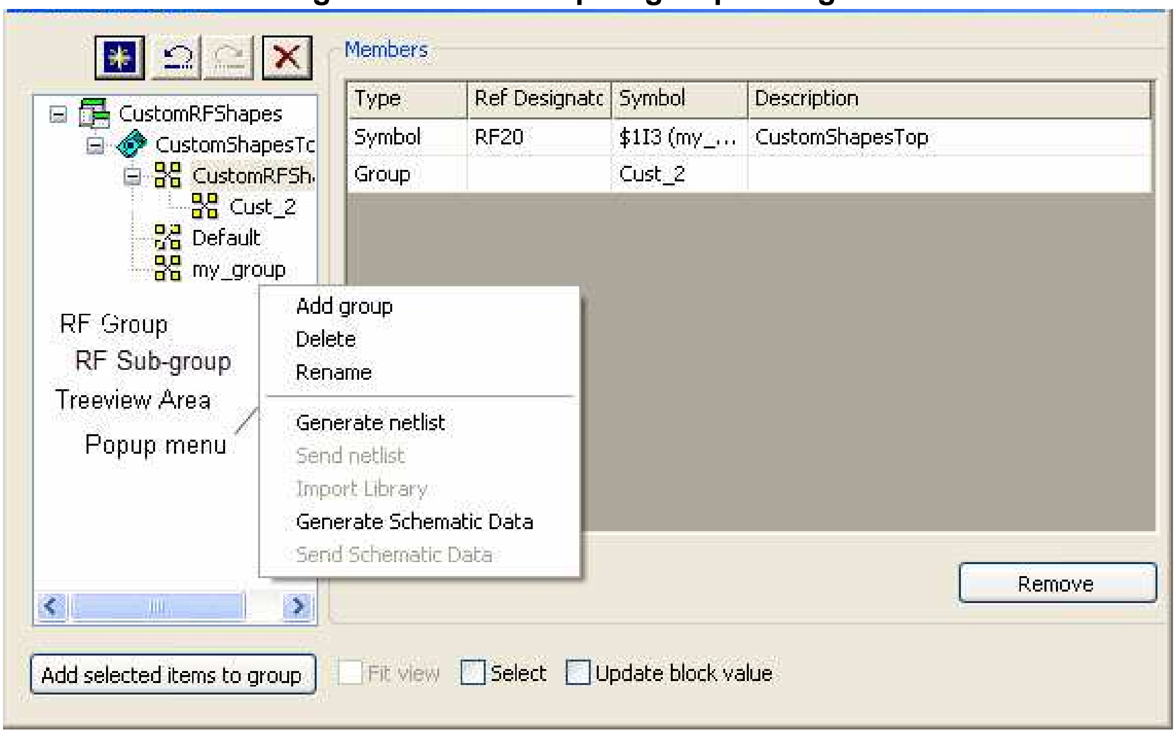


Table 2-5. RF Group/Ungroup Dialog Box Content

Field	Description
New	Adds a new RF group with the selected heading name with an appended number.
Undo	Restores the last deleted RF group with members.
Redo	Reverses the previous Undo.
Delete	Deletes the selected RF group and subgroups. Members of the RF group and subgroups are placed in the Default RF group.
Treeview area	Displays the project tree. This is identical to the Navigator window **RF Groups** tab (Navigator RF Groups Tab).
Members	Lists all the symbols within an RF group.
Remove	Click, removes selected members from the RF group and places them in the Default RF group. Use **Ctrl** or **Shift** to select more than one item.
Add selected items to group	Click, adds the selected symbol(s) to the active RF group.
Fit view	Checked, selected objects fit into the dialog box viewer.
Select	Checked, enables cross probing between the RF Group/Ungroup dialog box and the schematic capture tool.
Update block value	Checked, updates the block level value (RF Parameters Dialog Box - Schematic Capture). Unselected, updates only the instance value.

Table 2-6. RF Group/Ungroup Dialog Box Treeview Popup Menu Items

Selection	Description
Add group	Creates an RF group under the selected item in the treeview area.
Delete	Deletes the selected RF group. If there are members within the RF group, a confirmation dialog box appears. Selecting **Yes** deletes the RF group and subgroups. Members within the deleted RF groups become part of the Default RF group.
Rename	Enables renaming of the selected RF group.

Table 2-6. RF Group/Ungroup Dialog Box Treeview Popup Menu Items (cont.)

Selection	Description

Generate netlist	Creates a netlist, HTML, and xml file at the following locations:
	`<project directory>\RF\netlist\<project name>_<group name>_<current date (yyyymmdd)>_<current time (hhmmss)>.net`
	
	`<project directory>\RF\netlist\html\<project name>_<group name>_<current date (yyyymmdd)>_<current time (hhmmss)>.html`
	
	`<project directory>\RF\netlist\xml\<project name>_<group name>_<current date (yyyymmdd)>_<current time (hhmmss)>.xml`

Send netlist	Sends a schematic capture RF group netlist to the ADS RF Simulator (Sending Netlist). The dynamic link must be connected and an RF project needs to be open in the RF simulator (Connecting Schematic Capture to RF Simulator Server).
Import Library	Synchronizes the Central Library (CL) with the RF simulator library. If there is a difference, the RF simulator updates the CL (Adding ADS RF Shapes to Central Library and Adding AWR/MWO RF Shapes to Central Library).

Generate Schematic Data	Creates a *.iff* file (Interchange File Format) at the following locations:
	
	`<project directory>\RF\iff\<project name>_<group name>_<current date (yyyymmdd)>_<current time (hhmmss)>.iff`
	
	`<project directory>\RF\xml\<project name>_<group name>_<current date (yyyymmdd)>_<current time (hhmmss)>.xml`

Send Schematic Data	Sends a *.iff* file to the ADS RF Simulator or MWO RF Simulator (Sending Schematic Data). The schematic capture tool must be connected to the RF simulator with an active RF project (Connecting Schematic Capture to RF Simulator Server).

Related Topics

Creating an RF Group

Placing RF Symbols

Sending Schematic Data

Moving Members Between RF Groups

Adding an RF Varblocks

Sending Netlist

Creating an RF Group

When you create an RF group, you can send that group's netlist or schematic data to the RF simulator.

Use the following procedure to create an RF group.

Prerequisites

- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

1. Click **RF Group/Ungroup** ( ) in the RF toolbar.
2. Select a level in the treeview area.
3. Click **New** .
4. Select the new RF group.
5. Choose **Rename** from the popup menu and type in the new RF group name.
6. Select the RF and non-RF symbols on the schematic that you want to be part of the new RF group.
7. Click **Add selected items to group**.

Results

The selected symbols become members of the new RF group.

Related Topics

Moving Members Between RF Groups
Sending Schematic Data
Creating Variables in RF Shape Parameters
Sending Netlist
Adding an RF Varblocks
RF Group/Ungroup Dialog Box

Moving Members Between RF Groups

You can move group members between RF groups to simulate specific functions or areas within the RF design.

Use the following procedure to move members between RF groups.

Prerequisites

- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

1. Click **RF Group/Ungroup** ( ) in the RF toolbar.
2. Select the RF group that contains the members to be moved.
3. Select the members to be moved. Use **Ctrl** or **Shift** to select more than one item.
4. Drag-and-drop the selected members to the other RF group in the treeview area,

Results

The selected members move to the other RF group.

Related Topics

Creating an RF Group	Sending Netlist
Sending Schematic Data	Adding an RF Varblocks
Creating Variables in RF Shape Parameters	RF Group/Ungroup Dialog Box

Navigator RF Groups Tab

To access: Choose **View** > **Navigator** > **RF Groups** tab

Description

Use the Navigator **RF Groups** tab to view the RF design hierarchy (Figure 2-8). Popup menu selections differ depending on the hierarchy level you select (Figure 2-9).

Figure 2-8. Navigator Window - RF Groups Treeview Area

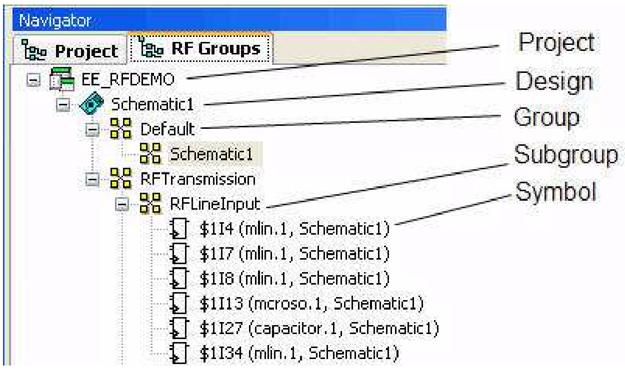
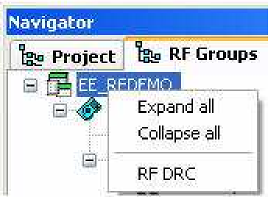
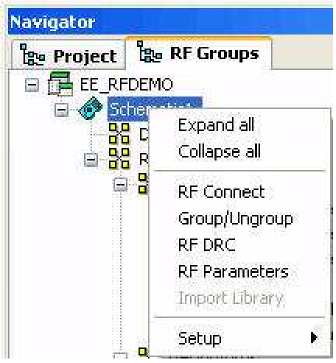


Figure 2-9. Navigator Window - RF Group Popup Menus

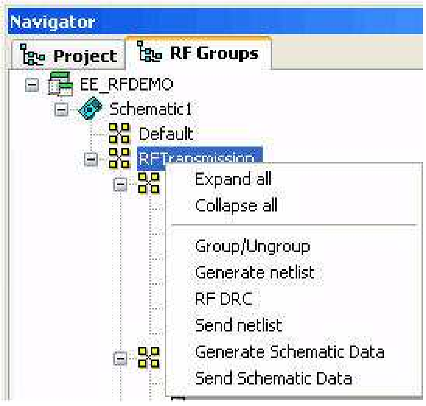
Project right-click menu



Design right-click menu



Group/Subgroup right-click menu



Symbol right-click menu

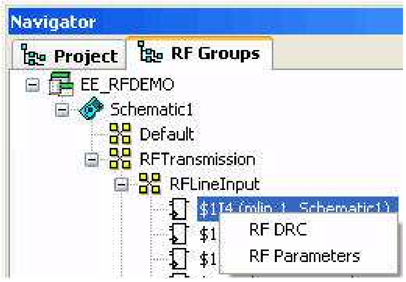


Table 2-7. Navigator Window - RF Groups Tab Treeview Popup Menu Selections

Selection	Description
RF DRC	Runs RF DRC on the selected RF group and displays the results in the **Output** > **RF DRC** tab window (RF DRC).
RF Connect	Opens the RF Connect Dialog Box (Schematic Capture).
Group/Ungroup	Opens the RF Group/Ungroup Dialog Box.
RF Parameters	Opens the RF Parameters Dialog Box - Schematic Capture.
Setup	Enables the following setup selections: • **Default Units** — Setting the RF Parameters Default Units of Measure • **Frequency Range** — Setting RF Frequency Range • **Substrates** — Adding a Substrate and Editing a Substrate
Generate netlist	Refer to Table 2-6.
Send netlist	Refer to Table 2-6.
Generate Schematic Data	Refer to Table 2-6.
Send Schematic Data	Refer to Table 2-6.

Related Topics

Creating an RF Group

Changing RF Symbol Parameters

Moving Members Between RF Groups

Connecting Schematic Capture to RF Simulator Server

RF DRC

Sending Netlist

To simulate an RF group netlist, you must send the netlist to the RF simulator.

Use the following procedure to send an RF group netlist to the RF simulator:

Prerequisites

- A dynamic link must exist between the RF server and the schematic capture tool (Connecting Schematic Capture to RF Simulator Server).
- A design must be open in the RF simulator.

Procedure

1. Click **RF Group/Ungroup** ( ) in the RF toolbar.
2. Select an RF group.
3. Choose **Send netlist**.

Results

The RF group appears in the RF simulator design.

Related Topics

Sending Schematic Data

Generate netlist

Moving Members Between RF Groups

Creating an RF Group

Generate Schematic Data

Navigator RF Groups Tab

Sending Schematic Data

To simulate RF group schematic data, you must send the schematic data to the RF simulator.

Use the following procedure to send schematic data to the RF simulator.

Prerequisites

- A dynamic link must exist between the RF server and the schematic capture tool (Connecting Schematic Capture to RF Simulator Server).
- A design must be open in the RF simulator.

Procedure

1. Click **RF Group/Ungroup** ( ) in the RF toolbar.
2. Select an RF group.
3. Choose **Send Schematic Data** in the popup menu.

Results

The RF group appears in the RF simulator design.

Related Topics

Sending Netlist	Creating an RF Group
Generate netlist	Generate Schematic Data
Moving Members Between RF Groups	Navigator RF Groups Tab

RF Parameters Dialog Box - Schematic Capture

Scope: DxDesigner

To access:

- Click **RF Parameters** () (RF Toolbar).
- Choose **View** > **RF** > **RF Parameters**.
- Choose **Navigator** > **RF Groups** tab > **Schematic**, choose RF Parameters.
- Click an RF DRC Error or Warning message (RF DRC).

Description

Use the RF Parameters dialog box to define the RF symbol parameters (Changing RF Symbol Parameters).

Figure 2-10. RF Parameters Dialog Box

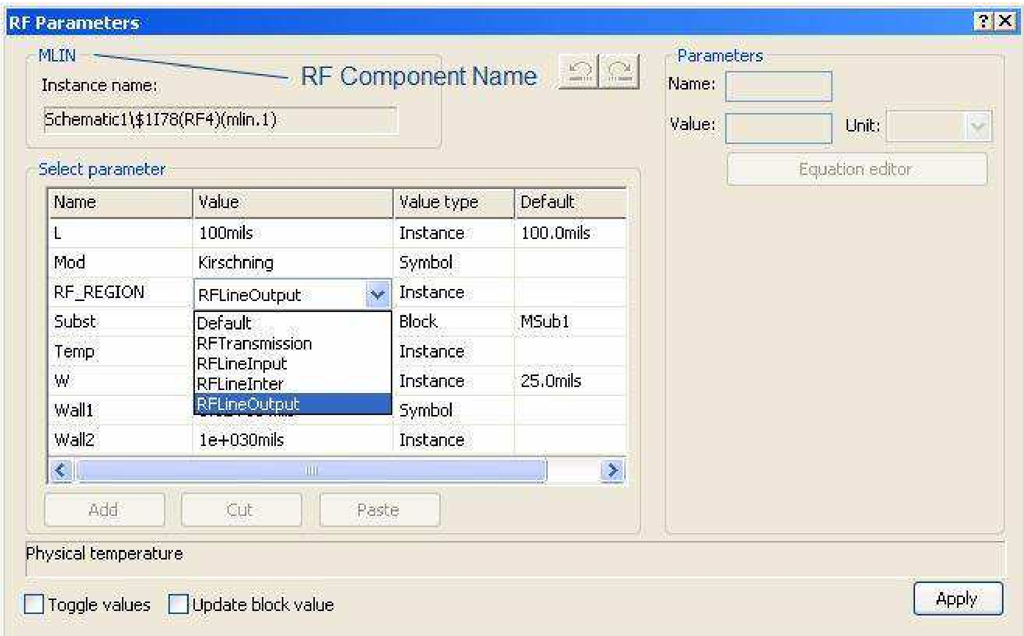


Table 2-8. RF Parameters Dialog Box Content

Element	Description
RF Component Name	Displays the name of the selected RF symbol.
Instance name	Displays the schematic number, instance name, and reference designator of the RF component.
Undo	Removes the latest changes in the select parameter field.
Redo	Reinstates the Undo changes.
Parameters section — Displays the selected parameter information.	
Name	Displays the name of the selected parameter.
Value	Defines the parameter value. Change the value as needed: • Real numbers require the format 0.0. • Evaluated expressions require parentheses "()". • Supported arguments are listed in Equation Editor Expressions.
Unit	Defines the parameter units. Choose from the dropdown list.
Equation Editor	Clicked, opens the RF Equation Editor Dialog Box (Figure 2-13 and Creating Variables in RF Shape Parameters).
Select parameter section — Lists the selected RF symbol parameters in the table.	
Parameters items	Lists the following RF symbol parameters: **Name** — The name of the parameter. **Value** — The value of the parameter. Use the dropdown arrow to display alternate selections. **Value Type** — Defines the level of the parameter value. • **Instance** — Parameter uses the instance level value. • **Block** — Parameter uses the block level value. • **Symbol** — Parameter uses the symbol level value. **Note**: The top-down hierarchical order for the Value type is: Instance, Block, Symbol.
Add	Click, adds the Equation Editor variable to the VARBLOCK.
Cut	Click, removes the selected variable from the VARBLOCK.
Paste	Click, reinstates the Cut changes.
Information Banner	Displays information about the selected parameter.
Toggle values	Checked, the numerical value appears in the Value field. Unchecked, the expression appears in the Value field.

Table 2-8. RF Parameters Dialog Box Content (cont.)

Element	Description
Update block value	Checked, updates the Block level value types. Unchecked, updates only the Instance level value types.

Related Topics

Adding an RF Varblocks

Adding an RF Varblocks

You can add an RF VARBLOCK to an RF group and create variables. Use VARBLOCK variables to define RF symbol parameters. Any change to a VARBLOCK variable automatically changes the variable in the RF symbol parameter.

Use the following procedure to add an RF VARBLOCK to an RF group.

Prerequisites

- A schematic capture project must be open.
- The RF toolbar must be displayed (RF Toolbar).

Procedure

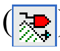
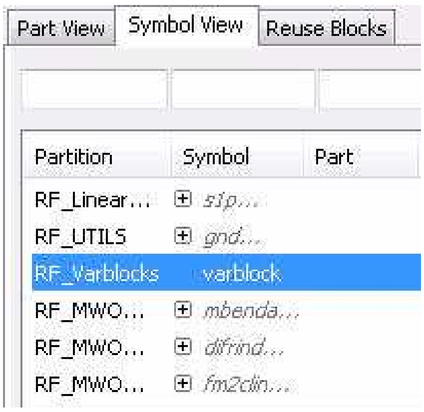
1. Select an RF group.
2. Click **DxDataBook** () in the DxDesigner toolbar**.**
3. Click the **CL View** tab.
4. Click the **Symbols View** tab and choose **RF_Varblocks** (Figure 2-11).

Figure 2-11. RF_Varblocks Selection



5. Place the VABLOCK symbol by dragging the symbol from the Place Symbol window into the drawing workspace and dropping it at the correct location. (Placing RF Symbols).

Results

The VARBLOCK symbol appears in the design and becomes a member of the selected RF group.

Related Topics

Creating Variables in RF Shape Parameters

Using Variables in RF Parameters

Creating an RF Group

RF Parameters Dialog Box - Schematic Capture

Equation Editor Expressions

Creating Variables in RF Shape Parameters

Create VARBLOCK variables so they define RF symbol parameters. When you change a VARBLOCK variable, you change all the RF symbol parameters that are defined with that variable.

Use the following procedure to create VARBLOCK variables.

Prerequisites

- The VARBLOCK is part of an RF group (Adding an RF Varblocks).

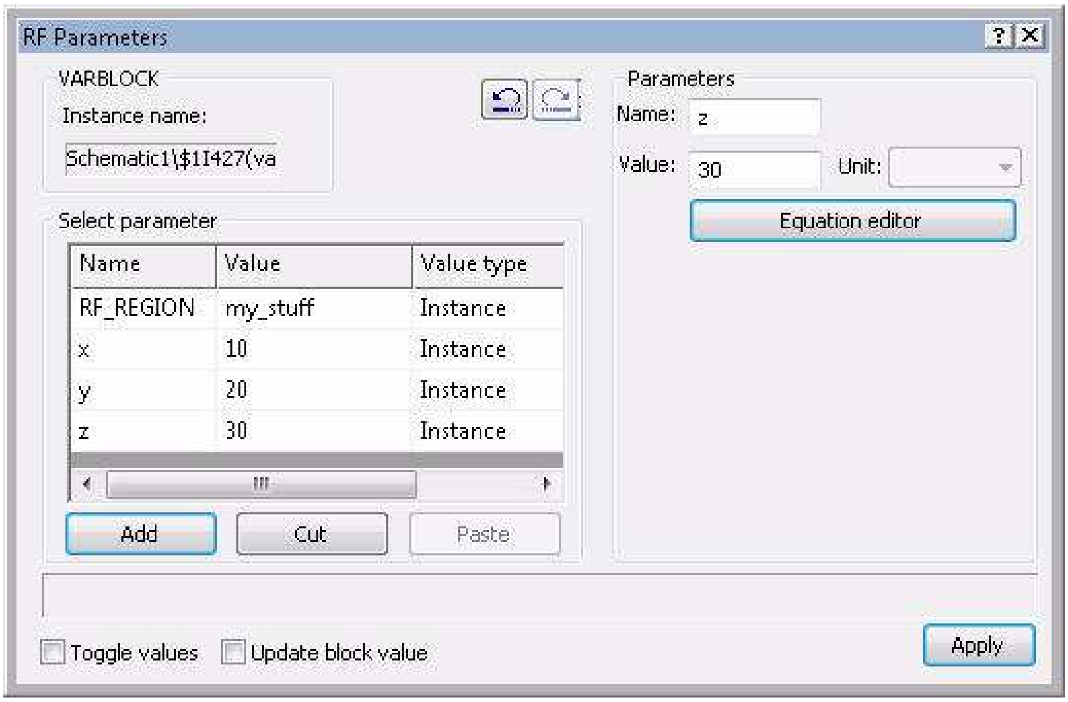
Procedure

1. Select the VARBLOCK.
2. Click **RF Parameters** () in the RF toolbar.
3. Type the variable name in the Name field (Figure 2-12).

Note

The Name text box accepts all alphanumeric characters (a-z, A-Z, 0-9) with no spaces.

Figure 2-12. RF Parameters Dialog Box - VARBLOCK



4. Type a value in the Value text box.
5. Click **Add**.

Results

The Select parameter section of the RF Parameters dialog box lists the new variables.

Related Topics

Using Variables in RF Parameters

RF DRC

Equation Editor Expressions

Changing RF Symbol Parameters

RF Parameters Dialog Box - Schematic Capture

Using Variables in RF Parameters

You can use variables to define RF parameters. When you change the variable, all of the RF parameters that are defined with that variable change accordingly.

Prerequisites

- The VARBLOCK is part of an RF group (Adding an RF Varblocks).

- The VARBLOCK contains variables (Creating Variables in RF Shape Parameters)

Procedure

1. Select the RF symbol.
2. Click **RF Parameters** () in the RF toolbar.
3. Select the parameter Value.
4. Click **Equation Editor**.

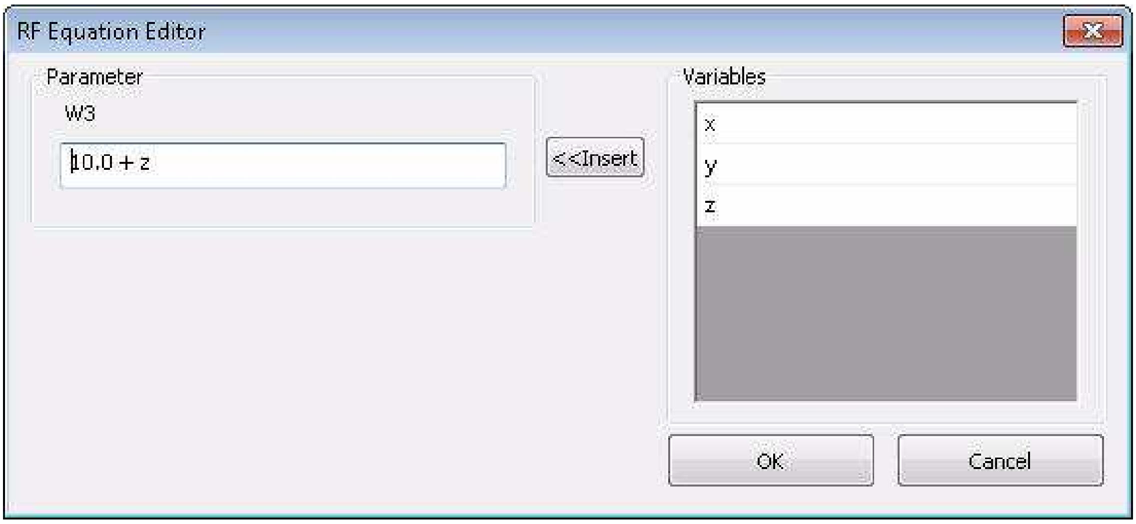
   Type the equation in the Parameter text box. In this example, the variable z was added to the Parameter text box (Figure 2-13).

   a. Select z in the Variables list.

   b. Click << **Insert**.

   In this example, the variable z was added to the Parameter text box (Figure 2-13).

Figure 2-13. RF Equation Editor Dialog Box - Variable



5. Click **OK**.

   The RF Equation Editor dialog box closes and the RF Parameters dialog box becomes active.

Results

The revised equation appears in the Value cell of the RF Parameters dialog box.

Related Topics

Creating an RF Group

RF DRC

RF Parameters Dialog Box - Schematic Capture

Changing RF Symbol Parameters

Equation Editor Expressions

Changing RF Symbol Parameters

You can change RF symbol parameters as needed to meet design requirements.

Use the following procedure to change the RF symbol parameters.

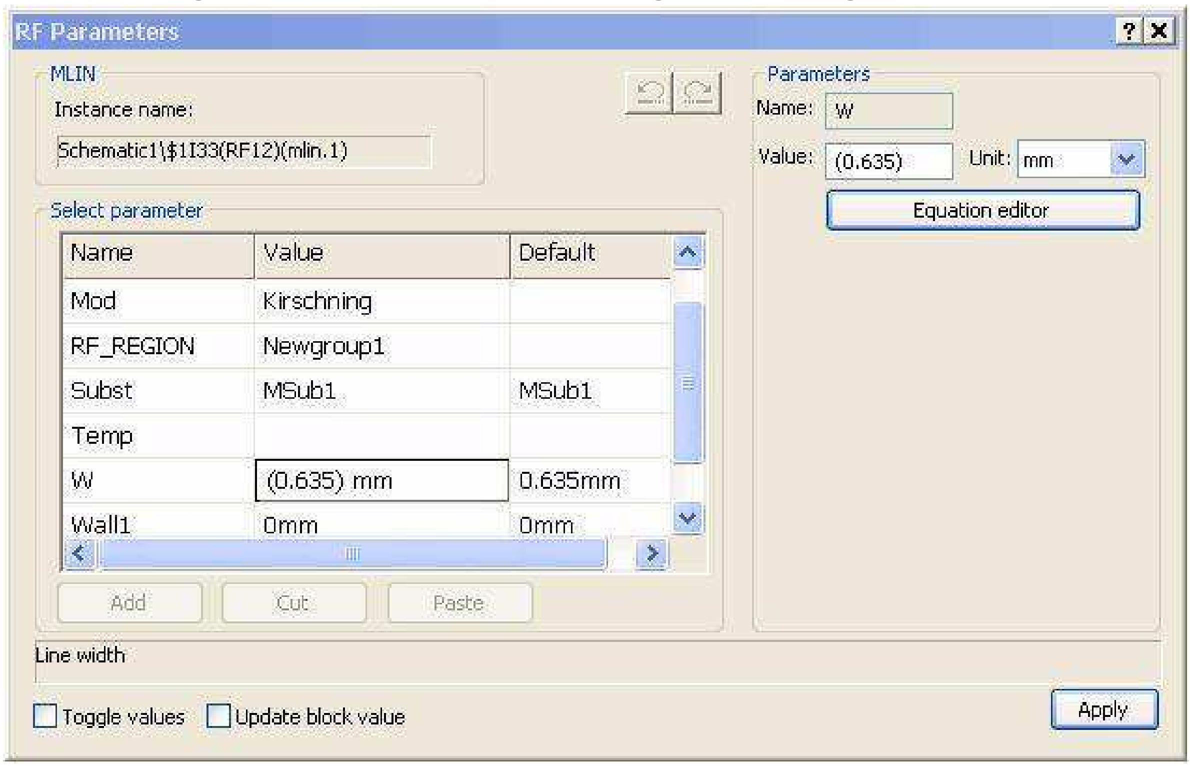
Prerequisites

- The RF design contains RF symbols (Placing RF Symbols).

Procedure

1. Select an RF symbol.
2. Click **RF Parameters** () in the RF toolbar.
3. Click the Value cell for the parameter you want to change (RF Parameters Dialog Box - Schematic Capture).

Figure 2-14. RF Parameters Dialog Box - Change Parameter



4. Under the Parameters section, enter a new value in the Value text box.
5. Select the appropriate unit of measure in the Units dropdown list.
6. Click **Apply**.

Results

The Select parameter table shows the new value for the parameter.

Related Topics

Sending Netlist

Sending Schematic Data

RF DRC

RF DRC

Design Rule Checking (DRC) applies to the RF group and its subgroups. RF DRC checks the dimensions of the RF symbol in conjunction with the RF frequency range.

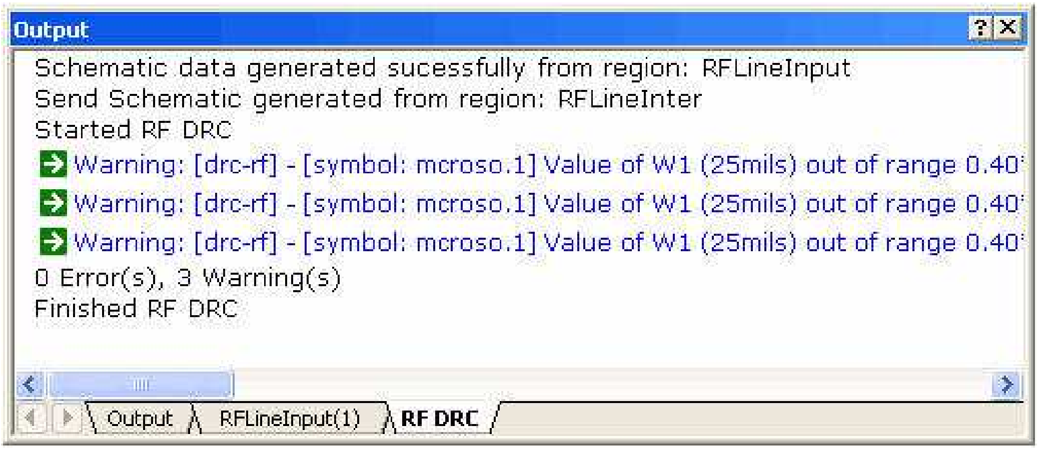
RF DRC checks the complete design. If you want to run RF DRC on a specific hierarchical level (Table 2-7), select the RF Group in the Navigator window, right-click and choose RF DRC from the popup menu.

Use the following procedure to run RF DRC.

Procedure

1. Click **RF DRC** () in the RF toolbar.
2. Select the **RF DRC** tab on the Output window (Figure 2-15).

Figure 2-15. RF DRC Output Window



3. Click any of the Error or Warning messages to highlight the offending symbol in the design.
4. Alter the RF design as needed to resolve the errors or warnings.
5. After resolving all of the errors and warnings, run RF DRC again to be sure everything is correct.

Results

If all of the DRC errors have been resolved, the Output window will report zero Errors and zero Warnings.

Related Topics

Sending Schematic Data	Sending Netlist
Changing RF Symbol Parameters	Navigator RF Groups Tab
RF Group/Ungroup Dialog Box	Setting RF Frequency Range

RF Connect Functionality

The RF connect functionality enables the schematic capture tool to connect to the RF simulator server. RF groups in the schematic capture tool are sent to the RF simulator with the dynamic link. Send the RF simulator changes back to the schematic capture tool to update the RF group (Connecting Schematic Capture to RF Simulator Server).

Related Topics

RF Toolbar	RF Connect Dialog Box (Schematic Capture)

Chapter 3
Forward Annotation

RF shapes are built in the layout tool as conductive shapes. RF shapes are modeled as a component and a cell. The Central Library (CL) contains a subset of the RF simulator RF shapes. Add RF shapes to the Central Library (Adding ADS RF Shapes to Central Library and Adding AWR/MWO RF Shapes to Central Library).

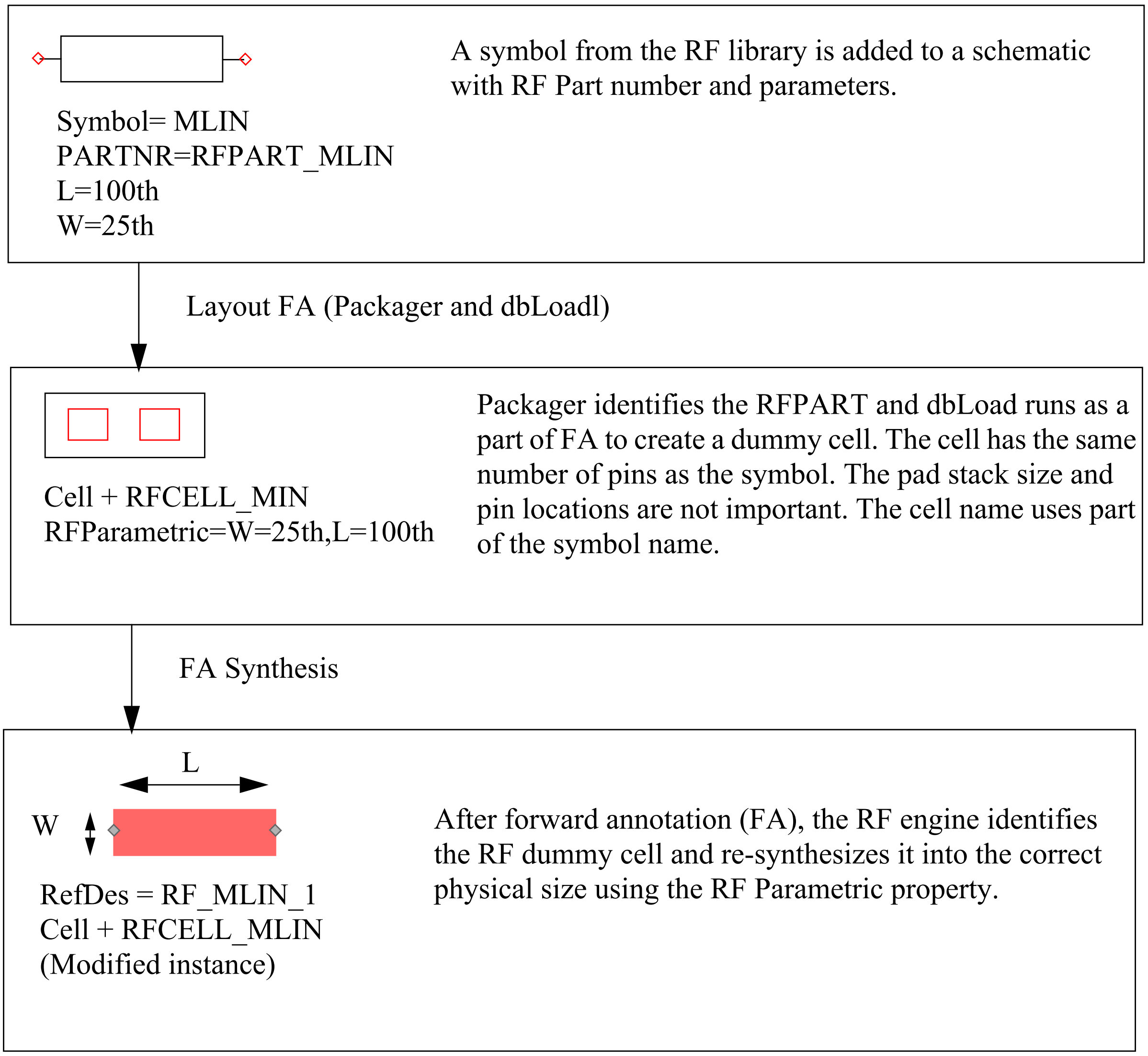
Each RF part has a part number RFPART_(ELEMENT_NAME), where ELEMENT_NAME is the RF Shape name. Choose **Tools > Package** to create a temporary cell name RFCELL_(ELEMENT_NAME), where ELEMENT_NAME is the RF Shape/Symbol name. The cell has the same number of pins as the symbol and is stored in the local design Central Library (Figure 3-1).

When you first place an RF shape in the schematic, the RFPART property is stored as part of the symbol. The reference designator (refdes) of the RF shape/symbol is the cell name. If two or more cells have the same cell name, the reference designators are incremental. For example, RF_MLIN_<*n*>, where *n* is an integer and ELEMENT_NAME is MLIN1, MLIN2 and so on.

On Forward Annotation (FA), the RF synthesis engine re-synthesizes the RF shape to its correct size based on the parameters of the schematic symbol. These parameters are stored as a property on the symbol named RF Parametric.

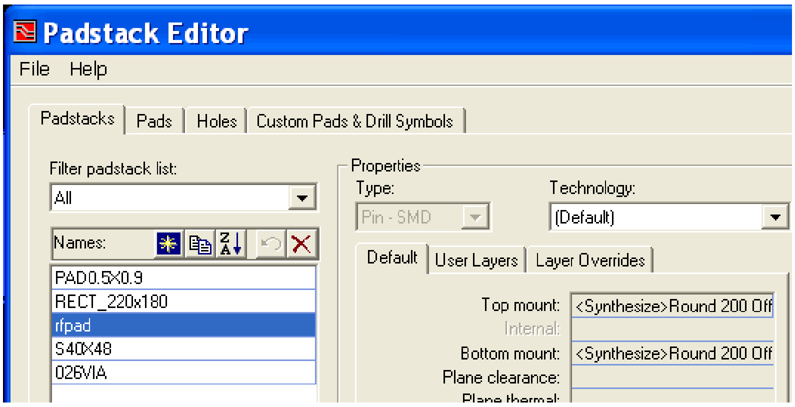
Forward annotate your design in schematic capture by choosing **Tools > Launch Expedition** or click **Expedition PCB** ( ). In Expedition PCB, select the RF Toolkit and then choose **Place > Place Parts and Cells** to view unplaced RF Groups.

Figure 3-1. Automatic Cell Creation in RF Flow



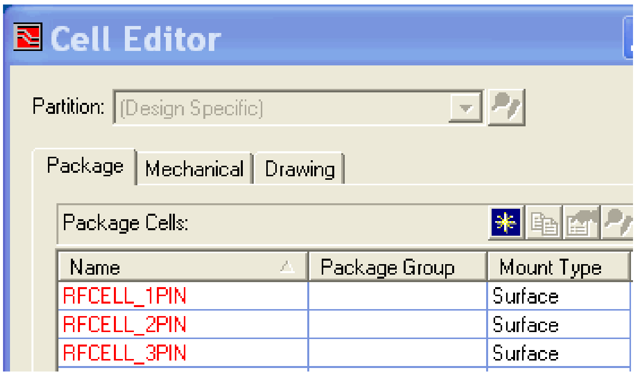
Setup > Padstack : Padstacks — The RF pad name for an RF cell and RF parts is rfpad. The rfpad is only available to the RF cells and RF parts (Figure 3-2).

Figure 3-2. RF Pad and Part Name - rfpad



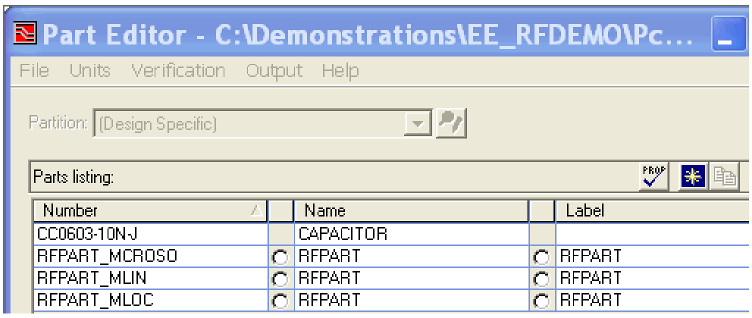
Setup > Cells Editor — RF cells are generic in nature and have a set number of pins. All pins are assigned to rfpad and are not editable (Figure 3-3).

Figure 3-3. RF Cell Pins Name



Setup > Parts Editor — The RF part name is unique to the part placed (Figure 3-4).

Figure 3-4. RF Part Name - Placed



Related Topics

Getting Started	RF Layout

Chapter 4
RF Layout

After you have Forward Annotated a schematic capture RF design you can place the RF parts from the layout tool, just as you would any other part.

This chapter covers the following topics:

Opening a PCB for RF Design

Before you can access an RF design in the layout tool you need an Advanced Technology Pro (RF) license. Acquire an Advanced Technology Pro (RF) license with one of the following methods:

- In the layout tool splash screen, check Advanced Technology Pro (RF).
- In the layout tool select **Setup** > **Licensed Modules** > **Advanced Technology Pro (RF)**.

When you have an Advanced Technology Pro (RF) license, the RF Toolkit () is selectable.

Related Topics

RF Layout

RF Toolkit Toolbar

Opening the RF Toolkit

The RF Toolkit enables you to create and modify RF designs in the layout tool.

To enable the RF Toolkit, click **RF Toolkit** in the Standard toolbar. You can then access the RF tools/commands with one of the following methods:

- Choose **Setup** > **RF**.
- The RF toolbar.
- RF from the workspace popup.

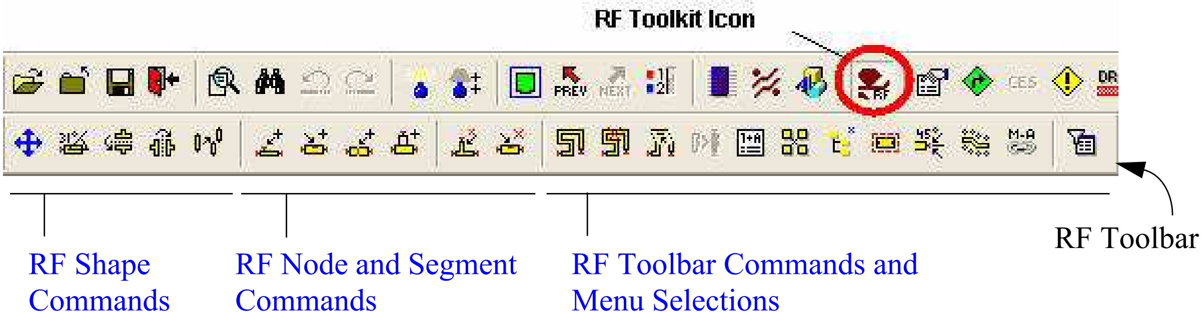
RF Toolkit Toolbar

The RF Toolkit enables you to create and modify RF designs in the layout tool. The RF Toolkit toolbar becomes active when you enable the RF Toolkit (Figure 4-1).

Note

You must have an Advanced Technology Pro (RF) license to enable and access the RF Toolkit.

Figure 4-1. RF Toolkit Icon and RF Toolbar



Related Topics

Creating and Modifying a Meander

RF Action Keys

Creating and Modifying RF Shapes

RF Layout

RF Shape Commands

The RF shape commands enable you to manipulate RF shapes in the layout tool. Table 4-1 lists the RF shape commands and operations.

Table 4-1. RF Shape Toolbar Commands

Command	Operation	Library shape	Meander
Move shape	Connecting RF Shapes by Moving	Yes	Yes
Angle shape	Rotating an RF Shape	Yes	Yes
Mirror about X	Mirroring an RF Shape	Yes	Yes
Mirror about Y	Mirroring an RF Shape	Yes	Yes
Copy	Copying a Meander	No	Yes

Related Topics

Creating and Modifying a Meander
RF Toolkit Toolbar
RF Action Keys
Creating and Modifying RF Shapes
RF Node and Segment Commands
RF Toolbar Commands and Menu Selections

RF Node and Segment Commands

The node and segment commands enable you to place, add, and delete nodes on RF shapes. Table 4-2 lists the RF node and segment commands and operations.

Table 4-2. RF Node and Segment Toolbar Commands

Command	Operation	Library Shape	Meander
Add edge node	Adding an Edge Node*	Yes	Yes
Add floating node	Adding a Floating Node*	Yes	Yes
Add segment	Adding a Segment	No	Yes
Add stub	Adding a Stub	No	Yes
Delete node stub	Deletes the selected node stub.	No	Yes
Delete floating node	Deletes the selected floating node.	No	Yes
* You can also define the node entry angle — RF Entry Rules Dialog Box			

Related Topics

Creating and Modifying a Meander
RF Toolkit Toolbar
RF Action Keys
Creating and Modifying RF Shapes
RF Shape Commands
RF Toolbar Commands and Menu Selections

RF Toolbar Commands and Menu Selections

The RF toolbar commands and menu selections enable you to create RF shapes and specify their properties and entry rules (Table 4-3).

Table 4-3. RF Toolbar Command and Menu Summary

Command	Description
Add Meander/Start Meander	Opens the Meander Properties Dialog Box. Use the Meander properties dialog box to define the meander properties.
Delete meander	Deletes the selected meander.

Table 4-3. RF Toolbar Command and Menu Summary (cont.)

Command	Description
Add Meander Segment	Adds a segment by changing the selected segment into two segments (Adding a Segment).
Edit Meander	Alters the angle of the selected meander and segment position (Editing a Meander).
Convert RF shape	Converts a conductive polygon to an RF shape. Refer to Creating a Custom RF Shape.
Parametric properties/RF Parameters	Opens the RF Shape Parameters Dialog Box in which you alter RF shapes and define Var block variable parameters.
Group/Ungroup	Opens the Group/Ungroup Dialog Box in which you alter RF groups containing RF shapes and non-RF shapes.
Control pane/RF Counterpane	Opens the RF Control Dialog Box in which you alter RF design hierarchy. You can also access RF group, segment, and node property values with the RF Design tab and view RF library shapes with the RF Library Shapes tab.
Clearance rules	Opens the Clearance Editor Dialog Box in which you define RF shape and segment clearances.
Group Clearance Rules (menu selection only)	Opens the RF Group Clearance Rules Dialog Box in which you define RF group level clearances for an RF shape to trace, pad, via, plane, and top and bottom solder mask.
Entry rules	Opens the RF Entry Rules Dialog Box in which you define the node entry angles for node-to-node and node-to-trace connections.
Place vias	Opens the Place Via Dialog Box in which you place stitching vias.
RF Connect	Opens the RF Connect Dialog Box (Layout) in which you connect the layout tool to an RF simulator for bidirectional data transfers.

Table 4-3. RF Toolbar Command and Menu Summary (cont.)

Command	Description
Selection Filter	Opens the Selection Filter Dialog Box in which you define selectable objects in the layout tool.

Related Topics

Creating and Modifying a Meander

Creating and Modifying Nodes

RF Node and Segment Commands

RF Action Keys

Creating and Modifying RF Shapes

Using RF Groups

RF Shape Commands

RF Toolkit Toolbar

RF Action Keys

The Action Keys at the bottom of the work area provide quick access to commonly used commands. Figure 4-2 shows the RF Action Keys.

Figure 4-2. RF Action Keys

Table 4-4. Action Keys Summary

Action Key	Description
F2 Place RF shapes	Opens the Place Parts and Cells dialog box with which you can place RF groups, RF shapes, and parts in your design.
F3 Route RF Shapes	Enables you to route RF shapes. The Action keys change as follows: **F2 Add Meander** — Opens the Meander Properties Dialog Box.
F4 Edit RF Shapes	Enables you to edit RF shapes.
F5 Stitch Vias	Opens the Place Via Dialog Box, with which you can place stitching vias.
F11 Group/Ungroup	Opens the Group/Ungroup Dialog Box, with which you can group design objects into RF groups.

Related Topics

Creating and Modifying a Meander

Creating and Modifying Nodes

RF Toolkit Toolbar

Creating and Modifying RF Shapes

Using RF Groups

RF Shapes

RF Shapes

RF shapes are conductive polygons used to connect parts or build other RF shapes.

All RF shapes in the library may be forward- and back-annotated between the schematic capture and layout tools. If you change RF shape parameters, the parameters are also updated when you perform forward- and back-annotation.

You define clearance rules for RF shapes with the Clearance Editor Dialog Box.

Related Topics

Creating and Modifying a Meander
Creating and Modifying RF Shapes
RF Standard Shapes
Meander
User-defined RF Shapes

RF Standard Shapes

An RF standard shape is a pre-defined RF shape. RF standard shapes are available from the library (Adding ADS RF Shapes to Central Library and Adding AWR/MWO RF Shapes to Central Library).

You can view RF library with the RF Control Dialog Box, RF Library Shape tab. Standard RF shapes consist of one or two nodes (connection points) and a segment. As an example, Figure 4-3 shows a one and two segment RF shape with two nodes. A node is a connection point on an RF shape (RF Shape Node Types).

RF standard shapes are listed in RF Design Tools User's and Reference Manual Supplement.

Related Topics

Creating and Modifying a Meander
Creating and Modifying RF Shapes
Meander
User-defined RF Shapes

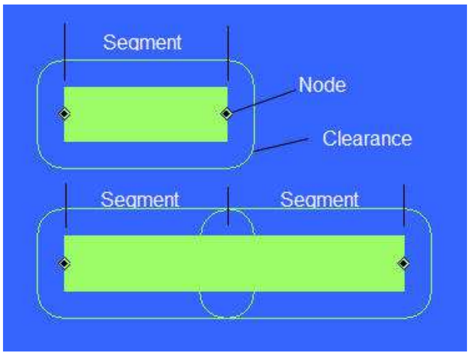
Meander

A meander is a segment or a series of segments of conductive RF shapes (Figure 4-3). Add a meander (RF shape) to connect parts or build a complex RF shape in layout.

Note

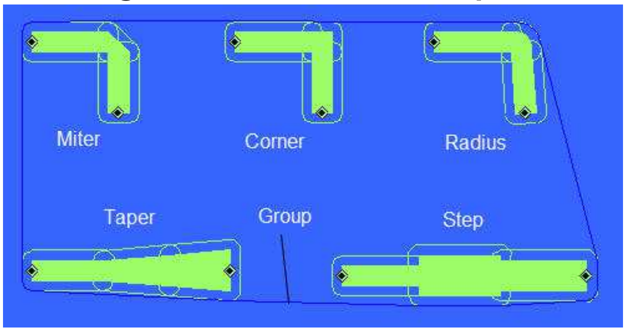
Meanders are not back-annotated to the schematic capture tool, and all meander nodes have the same net name.

Figure 4-3. Meander Segments



A meander contains at least one segment. You can create a meander with different corners, radii, and/or straight segments with varying widths (Figure 4-4).

Figure 4-4. Meander Examples



Related Topics

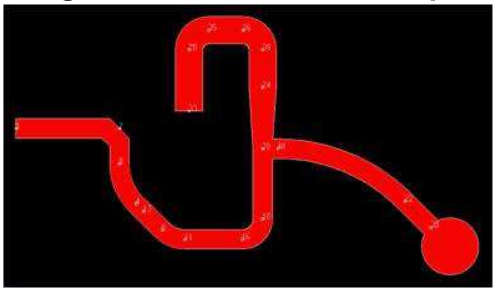
Creating and Modifying a Meander	Creating and Modifying RF Shapes
RF Standard Shapes	User-defined RF Shapes

User-defined RF Shapes

You can create a custom RF shape with standard RF shapes and meanders as shown in Figure 4-5 (Creating a Custom RF Shape).

You can add custom RF shapes to your library, which you can then place in the schematic capture tool (Creating a Custom RF Shape).

Figure 4-5. Custom RF Shape



Once a segment is part of an RF shape, you cannot move the segment independently of the entire RF shape.

When you create a custom RF symbol with schematic representation, the nodes are automatically mapped to pins on the schematic symbol.

Related Topics

Creating and Modifying a Meander

Creating and Modifying RF Shapes

Meander

RF Standard Shapes

Parametric RF Shape

A parametric RF shape has no fixed size or value. When you generate a meander or select an RF shape from the library, you create a parametric RF shape in your design. To access the parameters of an RF shape either:

- Double-click the RF Shape.
- Select the RF shape and choose **RF > RF Parameters** from the popup.

Related Topics

Creating and Modifying a Meander

Creating and Modifying RF Shapes

Meander

RF Standard Shapes

RF Shape Parameters

Each meander in an RF shape has specific parameters. You can view the RF shape parameters with the RF Control Dialog Box. The RF Design tab lists the selected RF shape parameters.

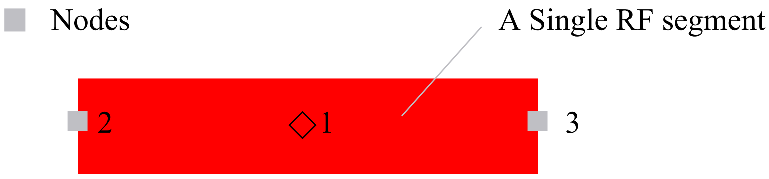
Changing Snap Points

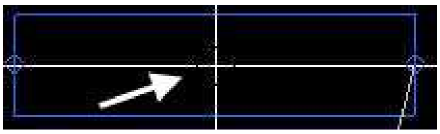
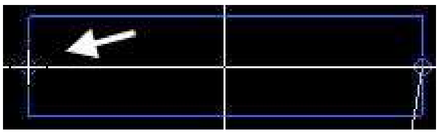
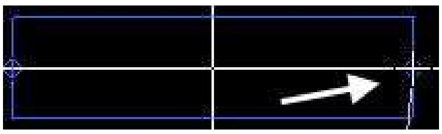
A snap point is a location on an RF shape to which the cursor attaches when you move an RF shape. The nodes of an RF shape are snap points, as is the center of the RF shape. When you select an RF shape, the cursor snaps to the center of the RF shape by default (Figure 4-6).

Procedure

- Change the snap point with the **TAB** key.

Figure 4-6. Snap Points on a Single Segment



	Initial selection Center snap point
	**TAB** Key, Left node snap point
	2nd **TAB** Key Right node snap point

Related Topics

Creating and Modifying a Meander

Connecting RF Shapes by Moving

RF Shapes

Creating and Modifying RF Shapes

Auto Arranger

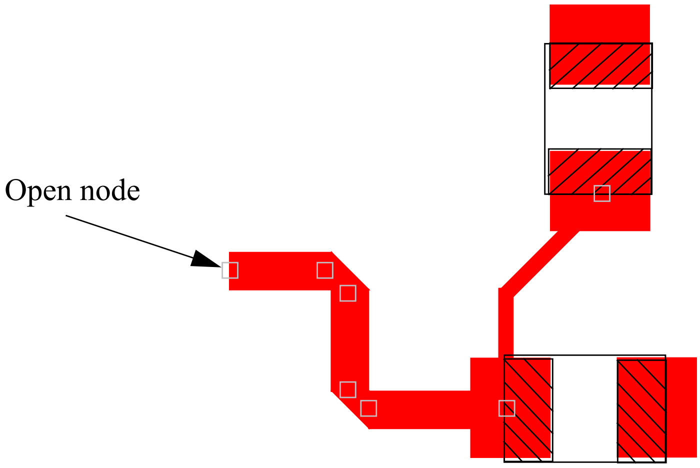
Open and Floating RF Shapes

Open and Floating RF Shapes

RF shapes with at least one unconnected node are called *open RF shapes* (Figure 4-7). RF shapes with every node connected are called *floating RF shapes*.

Note
DRC checks both open and floating RF shapes.

Figure 4-7. RF Shape - Open Node



Related Topics

Creating and Modifying a Meander

Creating a Meander

Changing Snap Points

Creating and Modifying RF Shapes

Changing an RF Net Name

RF Shape Node Types

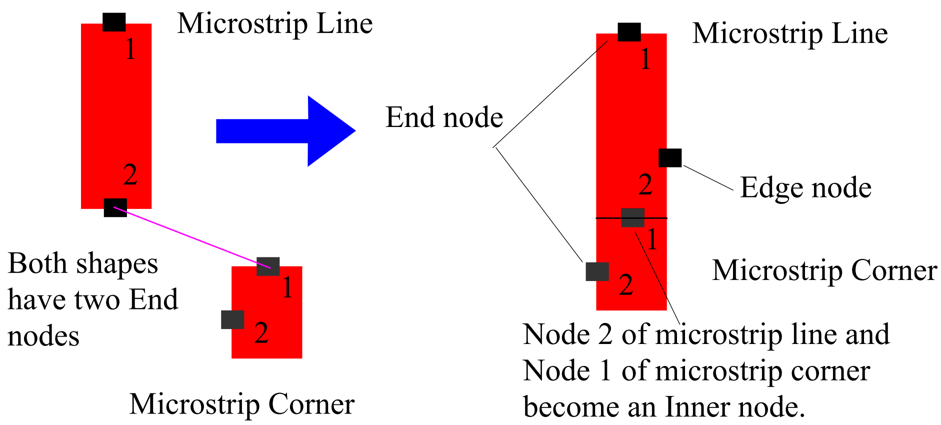
RF Shape Node Types

A node is a connecting point of an RF shape. An RF shape can have the following types of nodes:

- **Edge node** — A node placed on the edge of an RF shape (Adding an Edge Node).
- **End node** — A node that has only one adjacent segment.
- **Floating node** — A node placed on a library or custom RF shape that connects objects (Adding a Floating Node).
- **Inner node** — A node having two or more adjacent segments even if those segments belong to more than one RF shape.

When you connect RF shapes, End nodes can become Inner node as shown in Figure 4-8.

Figure 4-8. Inner Nodes



Related Topics

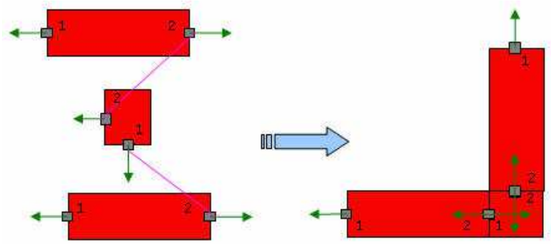
Setting RF Shape Node Entry Angle	Connecting Nodes and Pads with an RF Shape
Changing an RF Net Name	Creating and Modifying Nodes
Floating and Edge Nodes	Node-to-Node and Node-to-Trace Rules

Node-to-Node and Node-to-Trace Rules

Use the RF Control Dialog Box, RF Design tab to set the node-to-node and node-to-trace rules.

Figure 4-9 shows RF shapes with their connectivity rules (red) and the entry angle (green arrow). When you select RF shapes and choose **RF** > Auto Arranger, the selected RF shapes rotate and move to meet the entry and connectivity rules. In the example Figure 4-9 shows the results of the Auto Arranger (right side).

Figure 4-9. Node to Node Rules



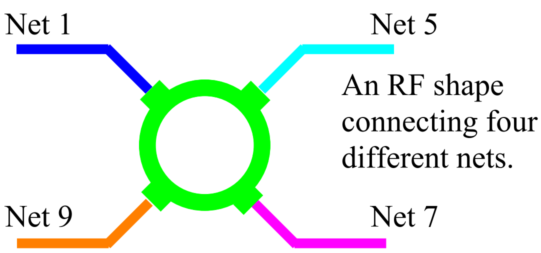
Related Topics

Setting RF Shape Node Entry Angle	Connecting RF Shapes by Moving
RF Shape Node Entry Angle	Creating and Modifying Nodes

Multiple Nets and Pad Connections

You can connect nets with an RF shape. Figure 4-10 shows an RF shape connecting four different nets, which creates an electrical RF net (Connecting Nodes and Pads with an RF Shape).

Figure 4-10. Connecting Multiple Nets



Related Topics

Creating a Custom RF Shape	Changing an RF Net Name
Connecting RF Shapes by Moving	Adding a Stub

Floating and Edge Nodes

Floating nodes and edge nodes are differentiated by their placements on an RF shape. Floating nodes can occur anywhere on an RF shape. Edge nodes can only occur at the edge of an RF shape. Both types of nodes enable connections to pads, traces, vias, and RF shapes. Refer to Adding a Floating Node and Adding an Edge Node.

Floating and edge nodes cannot move between RF shapes and they are not back annotated. When you add a floating or edge node to an RF shape, you can assign the node, node-to-node and node-to-trace rules (RF Control Dialog Box).

When you add a floating node, you assign the node to an existing schematic capture (or parent) node. The floating node is always associated with a schematic capture node within the RF shape. The node is never associated with a net. The parent node (schematic capture node), child node (floating or edge node), and RF shape always belong to the same net. When you assign a net to an edge node, it is also assigned to the floating node.

If you change a parent node name in the schematic capture tool, the floating node and/or edge node name assumes the parent node name during Forward Annotation (FA).

Additional information about floating and edge nodes:

- Parent nodes cannot be deleted in the schematic capture tool.
- The child node is associated with a parent node, which prevents floating or edge node changes.
- You cannot change connectivity in the layout tool.
- When you add a floating or edge node, you must define both net and entry rules to enable connection (Changing an RF Net Name and Setting RF Shape Node Entry Angle).
- When you push, rotate, or mirror an RF shape with floating nodes, the position and node name remain the same within the RF shape.
- When you update RF shapes that contain floating nodes, they are flagged as violations (RF Expired Nodes).
- Floating nodes in violation are reported until you manually change the node's Expired property in the RF Control Dialog Box pin properties.

Related Topics

Connecting Nodes and Pads with an RF Shape   Node-to-Node and Node-to-Trace Rules

Creating and Modifying Nodes

Mirror about X/Y

The RF Toolkit Toolbar mirroring commands (Mirror about X and Mirror about Y) enable you to mirror an RF shape or an RF group (a combination of RF shapes and non- RF shapes) about the X or Y axes (Mirroring an RF Shape and Adding an RF Group or RF Subgroup).

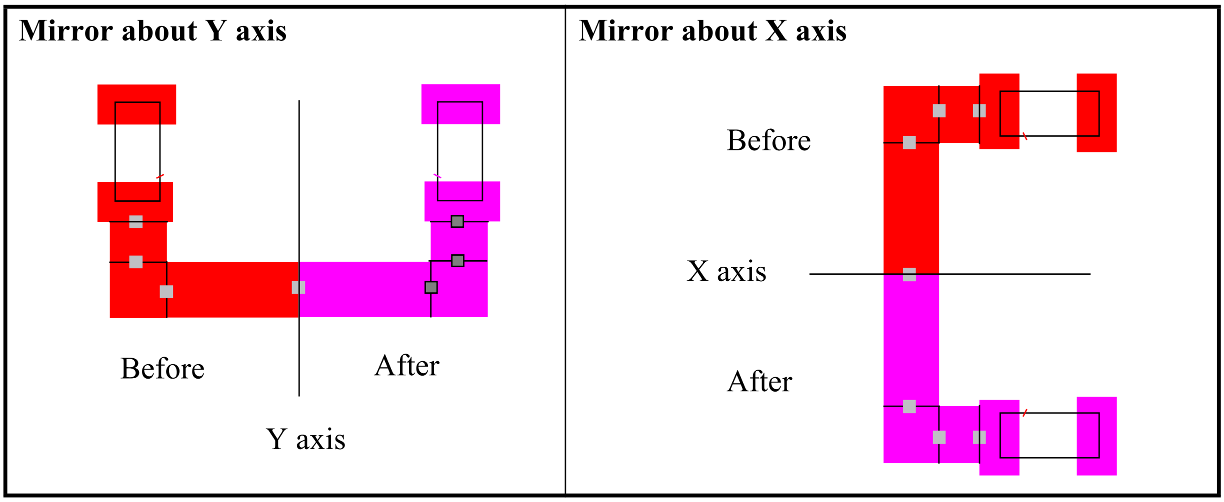
Non-RF parts in an RF group are mirrored only under the following conditions:

- The part must have two or more pins.
- The part must be a surface mounted or an embedded part.
- The part cannot be a through-hole part.
- The part pads must be square or rectangle.

Example 1

Figure 4-11 shows the mirroring about the X and Y axis at the snap point of the RF shape.

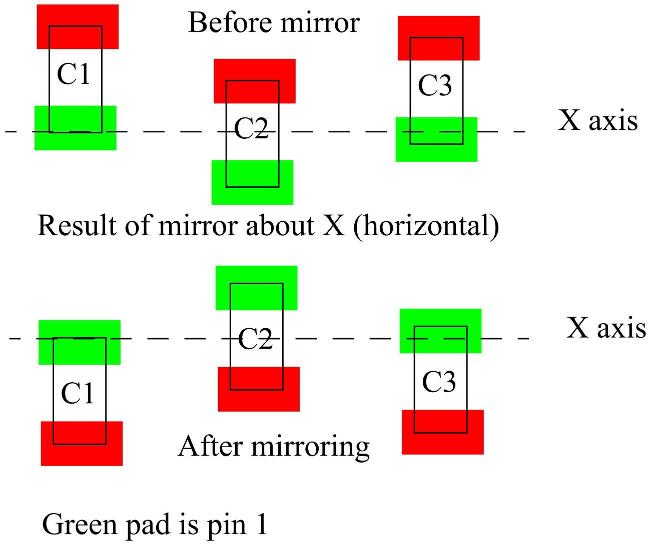
Figure 4-11. Mirror About X and Y Axis Examples



Example 2

Figure 4-12 shows C1 and C3 mirroring about the X axis, with the C2 center as the snap point. C1 and C3 maintain their relationship to C2 after mirroring.

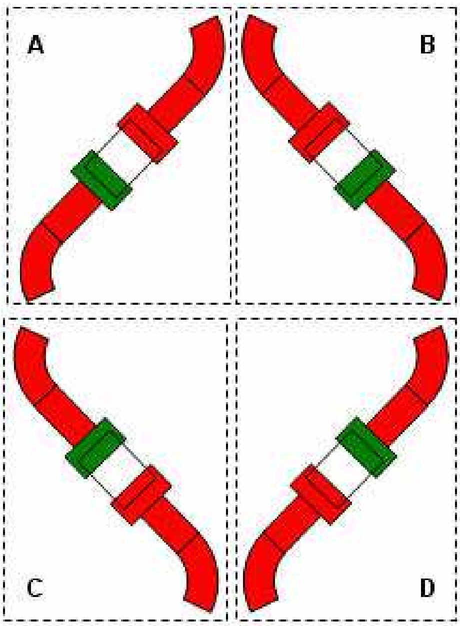
Figure 4-12. Mirror About X Axes - Object Center



Example 3

Figure 4-13 illustrates mirroring of an RF circuit placed at an odd angle.

Figure 4-13. Mirror of Angled Part



A. Starting position is 45°.

B. Mirroring A about the Y axis. Rotation is 135°.

C. Mirroring A about the X axis. Rotation is 225°.

D. Mirroring C about the Y axis. Rotation is 315°.

Related Topics

Creating and Modifying a Meander

Creating and Modifying RF Shapes

Moving RF Groups or Objects in RF Groups

Using RF Groups

Meander Properties Dialog Box

To access: Click **RF Toolkit** ( ) on the Standard toolbar, then do one of the following:

- Click **Add meander** ( ) on the RF toolbar
- **Setup > RF > Start Meander**
- Choose **RF > Start Meander** from the popup

Description

Use the Meander properties dialog box to define the meander properties.

Figure 4-14. Meander Properties Dialog Box

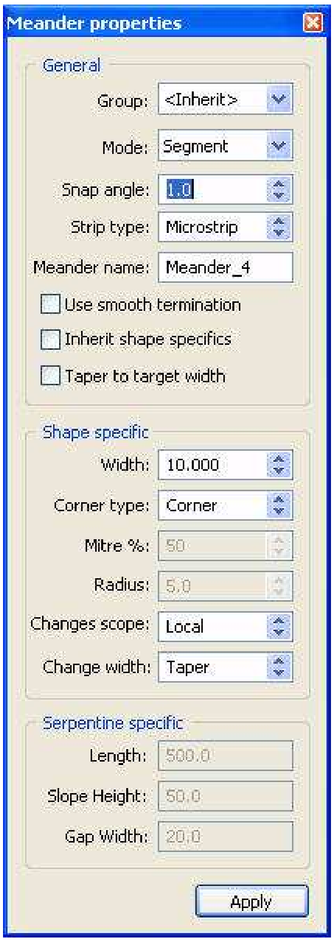


Table 4-5. Meander Properties Dialog Box Contents

Field	Description
General section	
Group	Defines all RF groups that are not fixed, locked, or frozen. • **<Inherit>** — Applies the RF group name of the selected object to the meander and uses the node of the selected object as a starting point (RF Group Contents). • **<Mouse Select>** — Applies the RF group name of the selected object to the meander. The meander does not need to be physically connected to the object. • ***<RF Group name>*** — Applies the active RF group name to the meander. The meander does not need to be physically connected to the RF group.
Mode	Defines the meander. • **Segment** — Defines a portion (segment) of a meander (the portion between mouse-clicks). • **Serpentine** — Defines a curved shape resembling the letter "s" (refer to the Serpentine field at the end of this table).
Snap angle	Defines the meander placement angle. For example, if you set the snap angle to 45°, the placement angles are 0°, 45°, 90°, 135°, and so forth.
Strip type	Defines the meander as a: • **Stripline** — A trace between ground planes. • **Microstrip** — A trace on the top or bottom layer of the PCB.
Meander name	Defines the meander name.
Use smooth termination when available	Checked, automatically creates an RF shape connection with the least amount of signal reflection (Meander Smooth Termination Connection). Unchecked, creates a meander connection with the current settings.
Inherit shape specifics	Checked, enables the meander to inherit properties from the selected RF shape. Selected shapes are: • **Meander** — Uses all RF shape-specific properties from the meander. • **RF shape** — Uses only width and layer properties to define the meander. • **Non-RF shape** — Uses only the layer property to define the meander. Unchecked, populates the Shape-specific section with default values.
Taper to target width	Checked, tapers meander width to the target width. Unchecked, enables you to attach the meander to different target widths.
Shape specific section — Defines the meander RF shape characteristics	
Width	Defines the meander width.

Table 4-5. Meander Properties Dialog Box Contents (cont.)

<table>
<tr><th>Field</th><th>Description</th></tr>
<tr><td>Corner type</td><td>Defines the meander corner type as:

• Free Radius — The largest possible radius for arbitrary meanders.
• Corner — A snap angle.
• Mitre — A mitre angle.
• Radius — An arc.

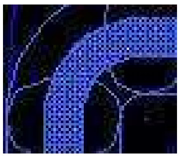
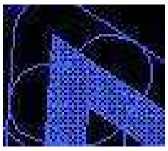
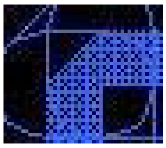
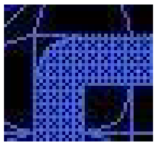
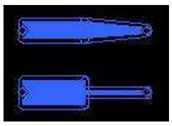
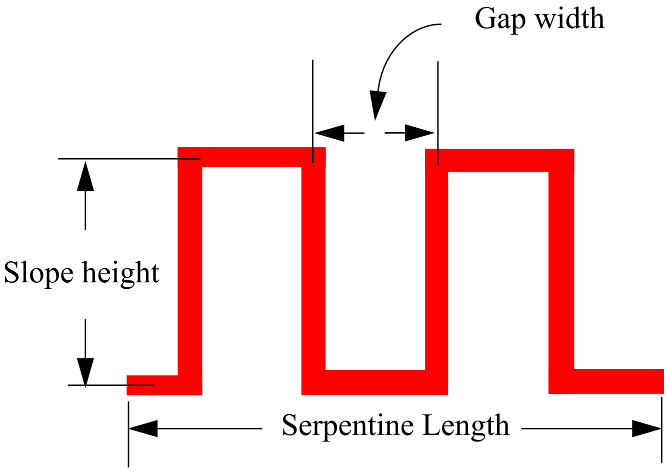
Free Radius Corner Mitre Radius
   </td></tr>
<tr><td>Mitre</td><td>(Mitre Corner type only) Defines the percentage of the outer corner to be removed.</td></tr>
<tr><td>Radius</td><td>(Radius Corner type only) Defines the radius of the arc applied to the outer corner.</td></tr>
<tr><td>Changes scope</td><td>Defines the meander scope.
• Local — Applies changes only to selected meander segments.
• Global — Applies changes to the entire meander.</td></tr>
<tr><td>Change width</td><td>Defines how the width change between segments:
• Taper — Gradual width change.
• Step — Abrupt width change.
</td></tr>
<tr><td colspan="2">Serpentine specific section — Defines the serpentine characteristics. Serpentine attributes are only available in the Serpentine mode (Mode field).

</td></tr>
<tr><td>Length</td><td>Defines the serpentine length.</td></tr>
</table>

Table 4-5. Meander Properties Dialog Box Contents (cont.)

Field	Description
Slope Height	Defines the slope height.
Gap Width	Defines the gap width.

Related Topics

Creating a Meander
Continuing a Meander on a Different Layer
Creating a Serpentine Meander
Editing a Meander

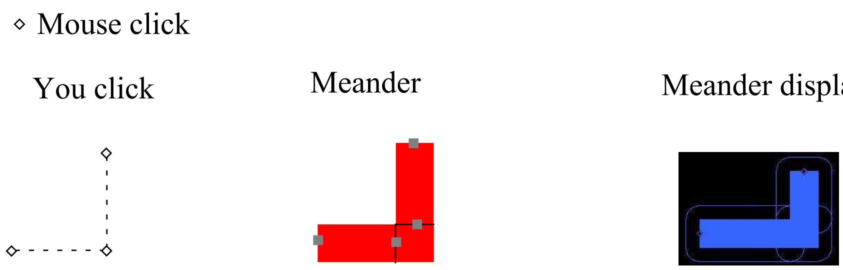
Meander Segment

Meander segments establish connections between nets and RF shapes as sell as between different RF shapes.

Note
RF shape characteristics and parameter values are described in the RF simulators documentation.

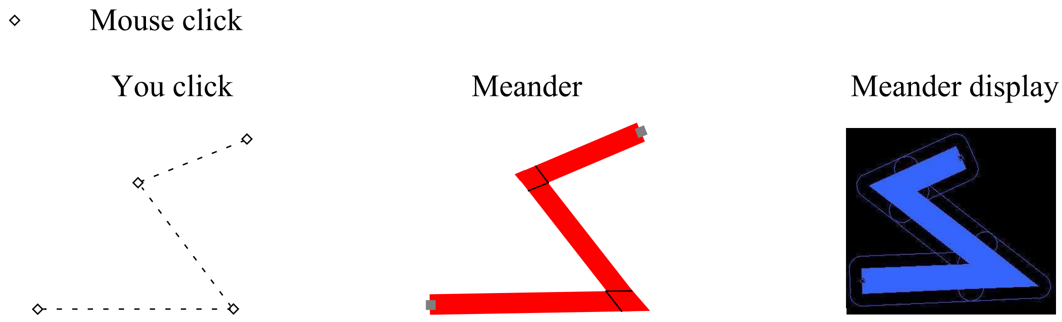
Figure 4-15 shows a simple meander with three segments. This example shows a **Corner type** of meander corner (Meander Properties Dialog Box).

Figure 4-15. RF Shape - Orthogonal Bend Construction



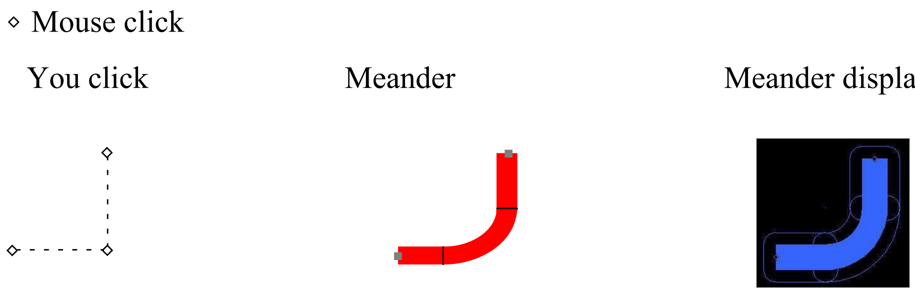
A non-orthogonal meander with the **Corner type** set to Corner or Mitre (Figure 4-16).

Figure 4-16. RF Shape - Non-orthogonal Bend Construction



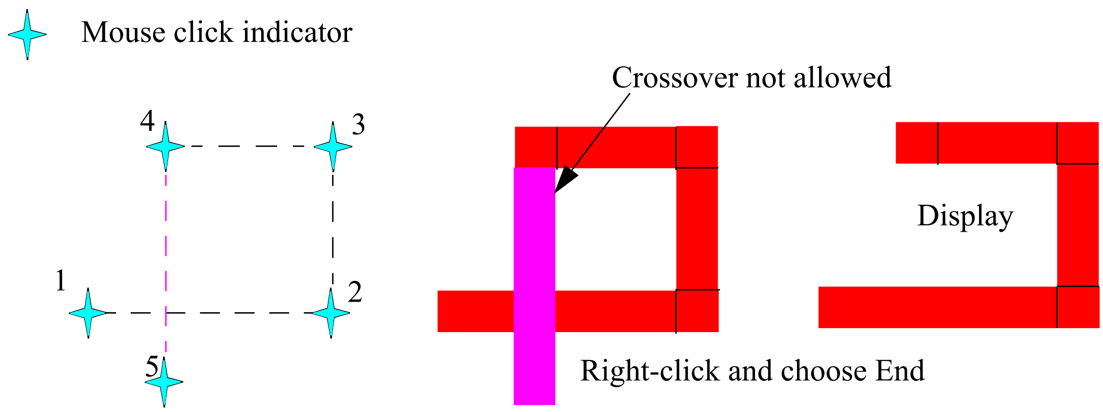
A meander with a **Corner type** of Free radius has curved corners (Figure 4-17).

Figure 4-17. RF Shape - Radius Bend Construction



The overlap restriction, detailed in the Overlap Check, prevents meander cross-overs (Figure 4-18)

Figure 4-18. Crossover Not Allowed



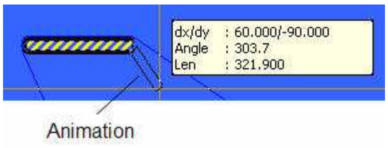
Related Topics

- Creating and Modifying a Meander
- Setting Clearances, Copying, and Creating Adjacent Layer Clearances
- Creating and Modifying Nodes
- Using RF Groups

Meander Display Length

When you add a meander, an animated border representing the RF shape appears during placement with a tag displaying the meander deviation, angle, and length (Figure 4-19).

Figure 4-19. Adding RF Shape



Related Topics

Creating and Modifying a Meander

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating and Modifying Nodes

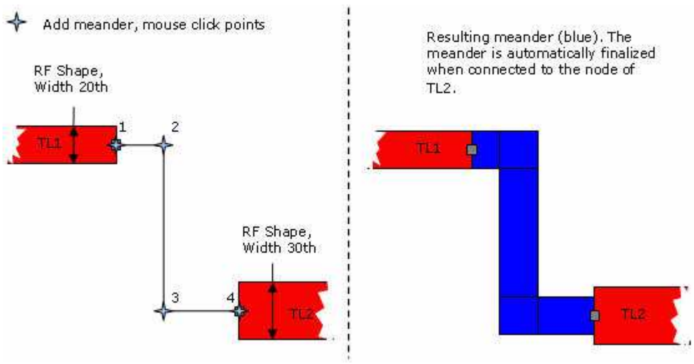
Meander Properties Dialog Box

Connecting RF Shapes with a Meander

When you connect RF shapes with a meander, the width of the meander can be defined by the RF shape starting node (Figure 4-20). The meander also inherits the net name from the starting node. The starting node and ending node (TL1 and TL2) have the same net name (Changing an RF Net Name).

Connect the RF shapes with a meander (Creating a Meander).

Figure 4-20. Connecting RF Shapes with a Meander



TL1 point 1 is the starting point of the meander in Figure 4-20. The meander uses the TL1 width If you check **Inherit shape specifics** on the Meander Properties Dialog Box, the meander is 20th, otherwise the width is the default value. When you click TL2 node (point 4), the meander completes. If you checked **Taper to target width**, the meanders width increases to 30th. In this example, Taper to target width was not checked.

Related Topics

Creating and Modifying a Meander

Creating and Modifying Nodes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Meander Properties Dialog Box

Meander Smooth Termination Connection

The **Use smooth termination** selection on the Meander Properties Dialog Box enables you to connect RF shapes automatically with the least amount of signal reflection. Figure 4-21 shows the meander as a simple S-curve connecting two RF shapes.

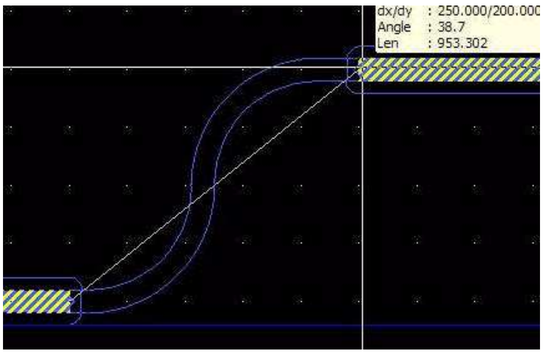
Use the following to create a smooth termination connection:

Procedure

1. Create a meander (Creating a Meander).
2. Check the **Use smooth termination** check box on the Meander Properties Dialog Box.
3. Set the other meander properties as necessary.
4. Click the start node on the RF shape and then click the end node.

   The smooth termination option automatically creates a meander as shown in Figure 4-21.

Figure 4-21. Smooth Connection



Related Topics

Creating and Modifying a Meander

Creating and Modifying Nodes

Automatic Meander Guide and Trim

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Meander Properties Dialog Box

Automatic Meander Guide and Trim

When you connect two nodes with a meander, automatic guidance and trimming occurs. The entry angle determines the guidance (RF Entry Rules Dialog Box). Trimming automatically removes slivers (meander overhang).

When you add a meander, the meander automatically aligns with the target shape (Selecting the Entry Angle During Meander Placement). Figure 4-22 and Figure 4-23 show guide and trim before you finalize the meander connection.

Note
Meander nodes must have the same net.

Figure 4-22. Meander Guide and Trim - Right Angle Placement

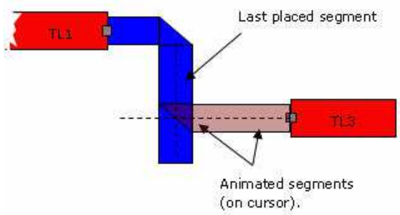
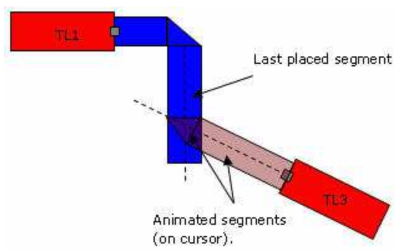
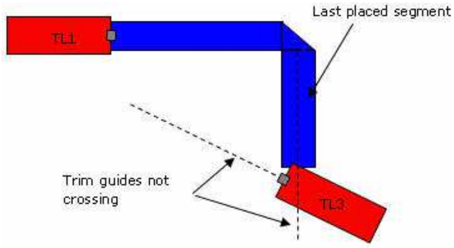


Figure 4-23. Meander Guide and Trim - Odd Angle Placement



The trim displays only if the two guide lines intersect as shown Figure 4-22 and Figure 4-23. The example in Figure 4-24, the guide lines do not intersect, which prevents any guide and trim presentation.

Figure 4-24. Meander No Guide and Trim - No Angle Placement



Related Topics

Creating and Modifying a Meander

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating and Modifying Nodes

Selecting the Entry Angle During Meander Placement

Meander Length Calculations

Meander Line-up Connections

Meander Properties Dialog Box

Meander Length Calculations

A meander's total length is the sum of each segment and corner. The corner type calculation depends on the corner characteristics. When you create a meander, use one of the following to calculate the corner length:

- Meander Corner Length Calculation
- Meander Radius Length Calculation
- Meander Mitre Length Calculation

Meander Corner Length Calculation

Corner length equation

$L = W \bullet S$

where:

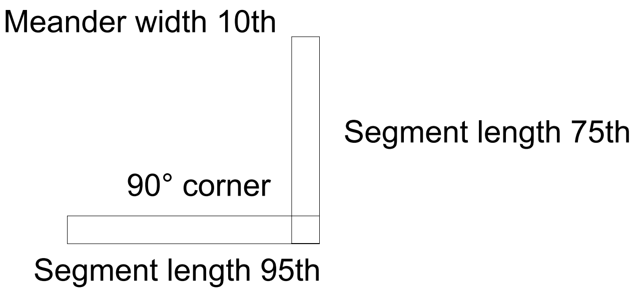
L = Corner Length

W = Corner width

S = $\sin\left(\frac{angle}{2}\right)$

angle = Degrees between segments

Example:



Corner effective length = 7.071th

Meander effective length = 95th + 75th + 7.071th = 177.071th

Meander Radius Length Calculation

Radius length equation

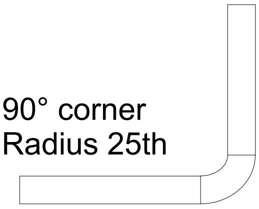
L = Angle • Radius

where:

L = Radius length
Angle = Degrees between segments
Radius = Radians value

Example:

Meander width 10th



$$\text{Radius effective length} = 90 \cdot 25 \cdot \frac{\pi}{180} = 39.270\text{th}$$

Meander effective length = 65th + 60th + 39.270th = 164.270th

Meander Mitre Length Calculation

The mitre corner length requires evaluation before you can calculate its length. Determine the mitre length with the following:

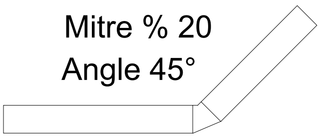
- Mitre Length Evaluation 1
- Mitre Length Evaluation 2

Mitre Length Evaluation 1

Mitre % $\leq S^2$
where:
S = $\sin\left(\frac{angle}{2}\right)$
angle = Degrees between segments

Use Meander Corner Length Calculation.

Example:



Meander and corner width 10th
Segment length 50 and 40th

Corner effective length = 3.8th
Meander effective length = 50th + 3.8th + 40th = 93.8th

Mitre Length Evaluation 2

Mitre % > S^2

where:

$\mathbf{S} = \sin\left(\frac{angle}{2}\right)$

angle = Degrees between segments

Mitre Corner equation

$$L = 2 \cdot \sqrt{a^2 + b^2 - 2(a \cdot b \cdot S)}$$

where:

L = Mitre corner length

$\mathbf{a} = d \cdot \left(\frac{\text{Mitre \%}}{S} - \frac{S}{2}\right)$

$\mathbf{b} = \text{Mitre \%} \cdot \frac{d}{2}$

$\mathbf{c} = \sqrt{1 - S^2}$

$\mathbf{d} = \frac{\text{Width}}{c}$

Width = Corner width

$\mathbf{S} = \sin\left(\frac{angle}{2}\right)$

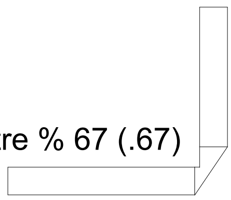
angle = Degrees between segments

Example:

Meander width 10th

Segment length 41.6th

Mitre % 67 (.67)



90° corner

Segment length 61.6th

a = 8.400000009

b = 4.737615436

c = 0.707106781

d = 14.14213563

Mitred effective length = 12.12023104th

Meander effective length = 12.12023104th + 61.6th + 41.6th = 115.320th

Related Topics

Creating and Modifying a Meander

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating and Modifying Nodes

Selecting the Entry Angle During Meander Placement

Meander Properties Dialog Box

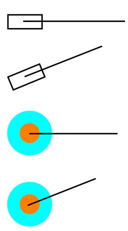
Meander Line-up Connections

The meander line-up enables you to use a specified entry angle to terminate a meander segment at a target object. This reduces slivers (meander overhang) at the target object (Automatic Meander Guide and Trim). The line-up connections occur between a part pad and a pin, a via, bond pad, or a part pad.

When you create a connection to a part pad to an object, the object's RF entry rules apply (RF Entry Rules Dialog Box). The RF entry rules define the meanders entry angle.

The via and bond pad use the Meander Properties Dialog Box Snap angle as an entry rule. With a meander property snap angle of 22.5º, every targeted via or bond pad has an assigned temporary RF entry rule of 22.5º. If the bond pad or via is rotated, the 0º entry rule matches the objects rotation as shown in Figure 4-25.

Figure 4-25. Meander Line-up for Bond Pad and Via



Bond pad rotation 0º and 0º entry rule

Bond pad rotation 22.5º and 0º entry rule

Via rotation 0º and 0º entry rule

Via rotation 22.5º and 0º entry rule

Related Topics

Selecting the Entry Angle During Meander Placement

Creating a Meander

Connecting a Pad to Pin with Meander Line-up

Automatic Meander Guide and Trim

Creating and Modifying a Meander

Use the following to create and manipulate meanders:

- Creating a Meander

- Editing a Meander
- Continuing a Meander on a Different Layer
- Creating a Serpentine Meander
- Selecting the Entry Angle During Meander Placement
- Connecting a Pad to Pin with Meander Line-up
- Meander Smooth Termination Connection

Related Topics

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating and Modifying Nodes

Using RF Groups

Creating a Meander

Meanders represent conductive material that connects RF shapes to each other and to nets and other non-RF shapes. Meanders have no representation in the schematic capture tool.

Tip: The RF simulator uses temporary netss to connect individual segments of a meander. These temporary nets are only used for RF simulation and are not utilized by the layout tool.

Note

If DRC prevents you from placing a meander in a confined area, use the **Alt** key to turn off DRC. After placing the meander, run Batch DRC to identify and correct clearance violations.

Note

Only the first and last segments of a meander use entry rules and can overlap an RF shape (Trace Connection Clearance and Node-to-Node and Node-to-Trace Rules).

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add meander** ( ).

   The meander is a member of the default group or the currently selected RF group (Group/Ungroup Dialog Box).
3. Set meander properties as needed (Meander Properties Dialog Box).

4. Click **Apply**.
5. Click a grid point, RF shape node, or snap point to start the meander.

   By default, meanders start on grid. To place a meander off grid, press the **Shift** key during placement. Meanders connect to RF shapes with nodes (not Net0):

   - Adding a Stub
   - Adding an Edge Node
   - Adding a Floating Node
6. Click a location and the current segment ends and starts a new segment.
7. Choose **End** from the popup or press **Esc** to terminate the meander. The **Esc** key does not place the last segment.

   The Meander properties dialog box remains active after you terminate a meander.
8. Place additional meanders as needed.
9. Close the Meander properties dialog box when you are finished creating meanders.

Results

A meander appears in your design and is part of an RF group.

Related Topics

Meander Length Calculations

Editing a Meander

Meander Properties Dialog Box

Continuing a Meander on a Different Layer

Editing a Meander

Use the **Edit Meander** ( ) command to alter the meander segment angle.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Edit Meander** ( ).
3. Select a meander segment.
4. Move the cursor to change the segment position.

   By default, the meander snaps to grid. To place the meander off-grid, press the **Shift** key during placement.

Note

If DRC prevents you from placing a meander in a confined area, use the **Alt** key to turn off DRC. After placing the meander, run Batch DRC to identify and correct clearance violations.

While editing a meander, choose an angle to define segment movement as shown in Figure 4-26. Angle Lock restricts meanders movement to the angles specified. Table 4-6 list the popup menu selections.

Figure 4-26. Meander Edit Popup Menu

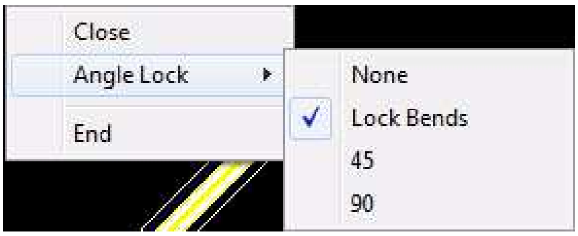


Table 4-6. Edit Meander Popup Menu

Popup Selections	Angle Lock Dropdown List	Selected Object Movement Description	*Cursor* Movement Direction
Close — closes popup			
Angle Lock — enables defined angle movement.	None	Object moves in any direction from its current position (default).	Any

Table 4-6. Edit Meander Popup Menu (cont.)

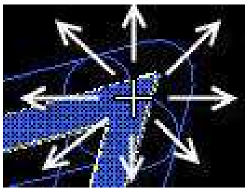
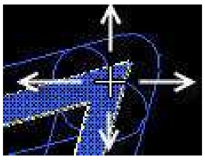
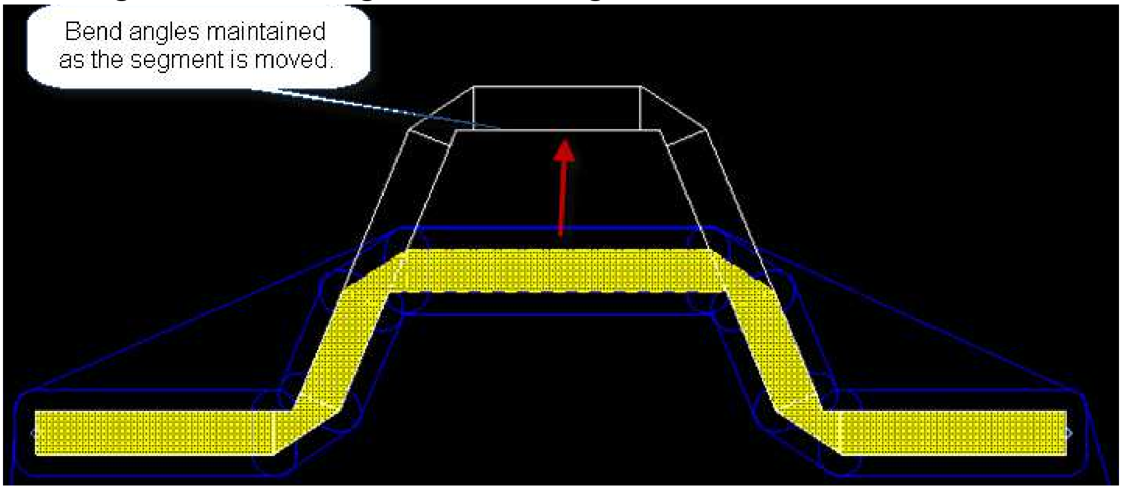
Popup Selections	Angle Lock Dropdown List	Selected Object Movement Description	*Cursor* Movement Direction
	Lock Bends	When you select this option, the angles at either side of the segment are maintained during the move. The length of the segment changes to accommodate the locked angles. See Figure 4-27.	Any.
	45º	Object only moves at 45º angles from its current position.	
	90º	Object only moves horizontally or vertically from its current position.	
End — terminates editing of the meander			

Figure 4-27. Moving a Meander Segment with Lock Bends Active



5. Move the meander to the new location.

6. Choose **End**.

Results

The meander changes position.

Related Topics

Continuing a Meander on a Different Layer
Adding a Segment
Creating and Modifying a Meander

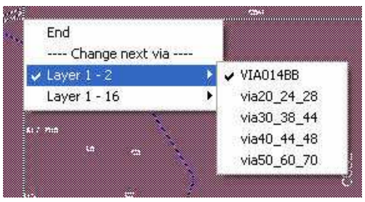
Continuing a Meander on a Different Layer

Meanders use RF vias to transition between layers (Vias in RF Designs). The RF via is included as part of the meander.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add meander** ( ).
3. Set meander properties as needed (Meander Properties Dialog Box).
4. Click **Apply**.
5. Start the meander (click and drag).
6. Place a via:
    a. Double-click to place a via. A via automatically appears in the layout tool.
    b. Choose the via layer range and RF via padstack from the popup menu (Figure 4-28).

Figure 4-28. Meander Via Selection



The RF via belongs to the same net as the meander nodes and is part of the meander in the RF group (Using RF Groups).

7. Continue creating the meander on the new layer.
8. Choose **End** to terminate the meander.

Results

A via connects meander segments on different layers.

Related Topics

Editing a Meander

Meander Line-up Connections

Creating a Serpentine Meander

Determining the best gap, height, and length of a serpentine meander for your design can be difficult. Create the serpentine in layout and send it to the RF simulator for evaluation. When you are satisfied with the results in the RF simulator, send the serpentine back to update the layout serpentine (Post Back to Schematic Capture and Layout or Posting Back MWO Updates to Layout).

Use the following to create a serpentine meander.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add meander** ( ).
3. Set the Mode field to Serpentine (Meander Properties Dialog Box).
4. Define the serpentine.
5. Make additional setting as needed.
6. Click **Apply**.
7. Select a meander start point by clicking in the design area. The start position is on a grid point or on a snap point. Use the **Shift** key during placement to select a point off-grid.

   The serpentine outline attaches to the cursor.
8. Click to place the serpentine.

   After placing the serpentine, the Mode field changes to Segment. If you want to place an additional serpentine, change the Mode field to Serpentine and click **Apply**.

Results

The serpentine meander becomes part of the RF design.

Related Topics

Creating and Modifying a Meander

Creating and Modifying Nodes

Selecting the Entry Angle During Meander Placement

When you connect a meander to a target (part pad, pin, via, or bond pad), you can define the entry angle.

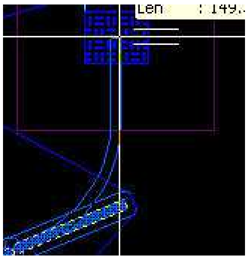
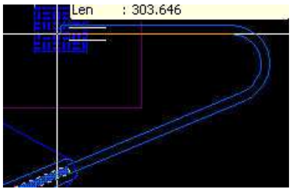
Prerequisites

- You are Creating a Meander and connecting it to a pad (end point).

Procedure

1. Place the cursor over the pad.
2. Click the **Tab** key to view each entry angle (Figure 4-29). In this example, the entry angle is 90º.
3. Click the pad when the desired entry angle displays.

Figure 4-29. Defining Entry Angle with the Tab Key

Entry Angle South	Entry Angle East
	

Related Topics

Creating and Modifying a Meander

RF Entry Rules Dialog Box

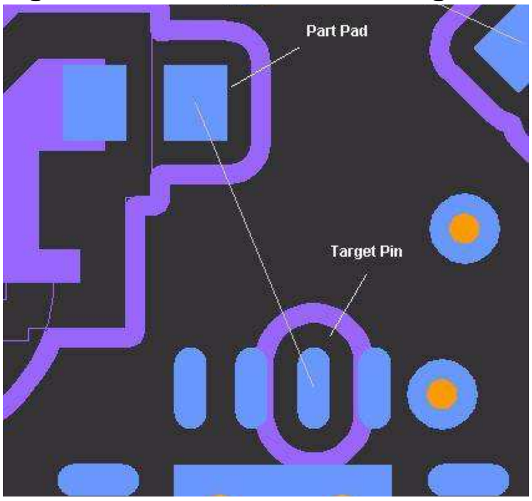
Connecting a Pad to Pin with Meander Line-up

Connect a meander from a part pad to a target (pin, via, or bond pad) with meander line-up (Meander Line-up Connections). The meander line-up enables you to select the connecting angle, reducing the meander overhang (slivers). A double-line at the target indicates meander line-up (Figure 4-31).

Prerequisites

- The target pin requires a defined entry angle (Setting RF Shape Node Entry Angle). In this procedure, the target pin has entry angles of 45º, 90º, and 135º (Figure 4-30).
- The part pad and target pin must have the same net name (Changing an RF Net Name).

Figure 4-30. Part Pad and Target Pin

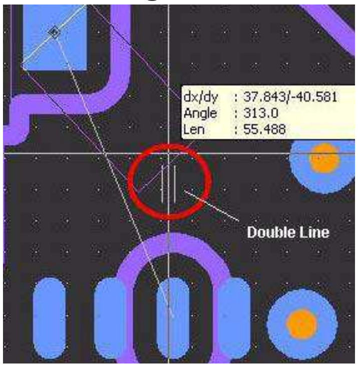


Procedure

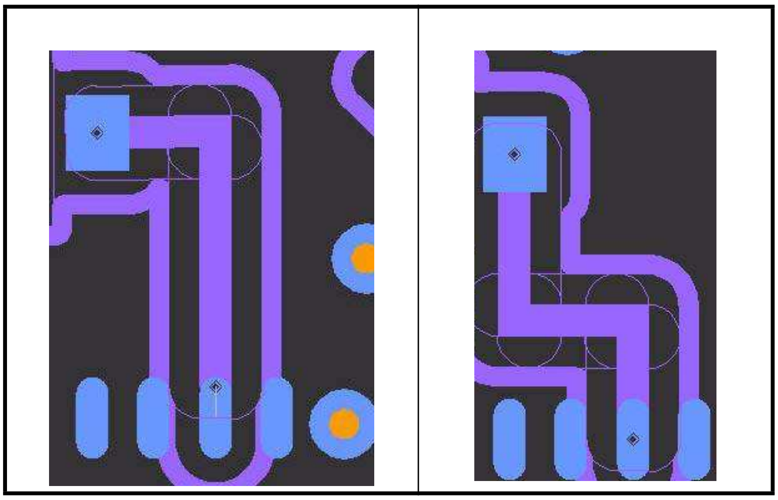
1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add meander** ( ).
3. Set meander properties (Meander Properties Dialog Box).
4. Start the meander by clicking on the Part Pad.
5. Move the cursor over the target pin (Selecting the Entry Angle During Meander Placement).

   A double-line attaches to the cursor. Move the cursor to the left and right and the double-line rotates 45º, indicating the other angles of entry. The double-lines only appears when the meander can enter the target with the defined entry angles.

Figure 4-31. Target Meander Line-up



6. Click to place the meander. In this example, two connection paths are possible.



Results

A meander connects the pad and pin with the desired target entry angle.

Related Topics

Creating and Modifying a Meander

Creating and Modifying Nodes

Creating and Modifying RF Shapes

Editing Existing RF Shapes

Use the following to edit and manipulate existing RF shapes:

Note
You cannot change the pin connectivity of an RF shape if it has been defined in the schematic capture tool.

- Adding a Segment
- Adding a Stub
- Connecting RF Shapes by Moving
- Rotating an RF Shape
- Mirroring an RF Shape
- Copying a Meander

Related Topics

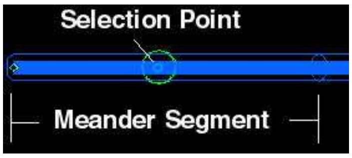
Creating and Modifying RF Shapes
Creating and Modifying a Meander
Creating and Modifying Nodes
RF Shapes

Adding a Segment

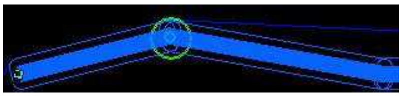
Add a meander segment to change the path of a meander.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add segment** ( ).
3. Select a meander segment (RF Shapes).
4. Place the cursor on the meander segment you want to alter.



5. Move the cursor, and the selected segment changes position.
6. Move the meander node point as needed, and click to place.



7. Press the **Esc** key to terminate the Add segment selection.

Results

A meander segment was added and the meander path changes.

Related Topics

Creating and Modifying a Meander | Editing Existing RF Shapes

Adding a Stub

The **Add Stub** command enables you to start a meander from the edge of a meander.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add stub** ().
3. Select the meander.
4. Place the stub on the edge of the meander.

   The Add edge node window appears (Adding an Edge Node).
5. Select **Node/Net** from the dropdown list.
6. Click **Apply**.

   The Meander Properties Dialog Box appears, enabling you to define the stub meander.
7. Create the meander (Creating a Meander).

Related Topics

Connecting RF Shapes by Moving

Creating and Modifying Nodes

Editing Existing RF Shapes

Connecting RF Shapes by Moving

Connect RF shapes by moving their open nodes into proximity. The open nodes become the snap points for the RF shapes connection (Changing Snap Points).

Prerequisites

- The connecting nodes must have the same net name (Changing an RF Net Name).
- The connecting nodes, Connect-to-plane properties must have a Connect value (RF Control Dialog Box).

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF shape.
3. Press the **Tab** key to make the open node the snap point.
4. Move the RF shape snap point to the other open node on the target RF shape. When the RF shape snap point is in proximity of the other open node, the RF shapes connect.

Related Topics

Creating and Modifying Nodes

Editing Existing RF Shapes

Auto Arranger

Rotating an RF Shape

An RF shape rotates in 0.01° increments around any RF shape open node.

Use the following procedure to rotate an RF shape.

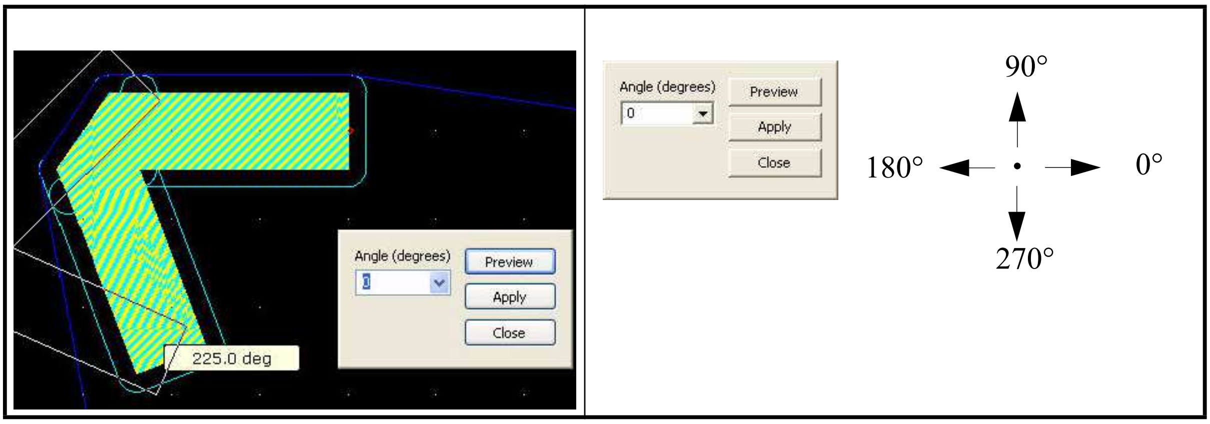
Prerequisites

- The RF shape must have an open node for the rotation point (RF Shape Node Types). If there is no open node, the RF Shape center becomes the rotation point.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select **Angle shape** ( ).
3. Select an RF shape.
4. Press the **Tab** key to select the desired snap point. The RF shape rotates around the snap point.
5. Rotate the RF shape around the snap point with the cursor. Observe the Angle tag and Angle window (Figure 4-32).
   a. Select an angle with the dropdown arrow and click **Preview**.
   b. Click **Apply**.
   c. Click **Close**.
6. Click to place the RF shape or specify the angle.

Figure 4-32. RF Shape Rotation



Related Topics

Mirroring an RF Shape
Editing Existing RF Shapes
Creating and Modifying RF Shapes

Mirroring an RF Shape

Mirror an RF shape or RF group around the X/Y axes. The RF shape or RF group mirror on the same layer or multiple layers (Mirror about X/Y).

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Select the RF shape or RF group.
3. Click the mirroring type:
   - **Mirror about X** ().
   - **Mirror about Y** ().

Related Topics

Creating and Modifying RF Shapes
Editing Existing RF Shapes

Copying a Meander

The RF **Copy** command enables you to copy an existing meander and make it part of your design.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select the meander.
3. Click **Copy** ( ).

   A copy of the meander attaches to the cursor.
4. Move the meander to a new location.
5. Click to place the meander.

   In the Group/Ungroup Dialog Box member list, the default name of the copied meander is “copy_of_<*Meander x*> with a default description <*RFPART*>_copy_of_<*Meander x*>”, where x is the copy number.

   All nodes of the copied meander are assigned to (Net0). The copied and original meander belong to the same RF group. Use the RF Control Dialog Box to change the net names (Changing an RF Net Name).
6. Add the copied meander to a different RF group (Adding Members to an RF Group).

Related Topics

Creating and Modifying a Meander

Editing Existing RF Shapes

Moving Members Between RF Groups

Creating and Modifying Nodes

Use the following to create, rename, and define entry angles for nodes:

- Adding a Floating Node
- Adding an Edge Node
- Changing an RF Net Name
- Setting RF Shape Node Entry Angle

Related Topics

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

RF Shapes

Auto Arranger

Adding a Floating Node

Add a floating node on an RF shape to connect the RF shape to traces, vias, part pads, and bond pads.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Add floating node** ().
3. Select an RF shape.
4. Click the location for the floating node.
5. In the Add floating node dialog box, select the node name from the Node/Name dropdown list.
6. Click **Apply**.

Related Topics

Creating and Modifying a Meander

Multiple Nets and Pad Connections

Creating and Modifying Nodes

Adding an Edge Node

Adding an Edge Node

Add an edge node to an RF shape to connect the RF shape to traces, vias, and pins. Edge nodes can only be placed on the edge of an RF shape (Floating and Edge Nodes).

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select the RF shape.
3. Click **Add Edge Node** ( ).
4. Click the location for the edge node.
5. In the Add edge node dialog box, select a net from **Node/Net** dropdown list.
6. Click **Apply**.

Related Topics

Creating and Modifying a Meander

Multiple Nets and Pad Connections

Creating and Modifying Nodes

Adding a Floating Node

Changing an RF Net Name

You can change the RF net name when the meander and pin require the same net name. However, you only perform this procedure if the RF elements are *not* schematic driven.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Control pane** ( ).
3. Choose RF Design tab.
4. Select the RF shape node.
5. Choose the net name from the net Value dropdown list in the property field.
6. Click **Apply**.

Related Topics

Creating and Modifying a Meander
Creating and Modifying Nodes
Using RF Groups
Auto Arranger

Setting RF Shape Node Entry Angle

Set the node-to-node and node-to-trace entry angle for a node with the RF Entry Rules Dialog Box.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Entry rules** ( ).
3. Select the part and node.
4. Define the node-to-node and node-to-trace node entry angles with the RF Entry Rules Dialog Box.

Results

New entry angles display or none in the case of Free.

Related Topics

Editing Existing RF Shapes

Auto Arranger

Creating and Modifying Nodes

Creating and Modifying RF Shapes

RF Control Dialog Box

To access:

- With the RF Toolkit () active, click **Control pane** () (RF Toolkit Toolbar)
- **Setup > RF > RF Control pane**
- Choose **RF > RF Control pane**

Description

Use this dialog box to view and change RF group, segment, and node property values.

The dialog box has the following tabs:

RF Control Dialog Box - Library Shapes tab — Displays all the RF shapes in the library. Select an RF shape to populate the property and preview fields. You can only view the library RF shapes and properties, you cannot edit them.

RF Control Dialog Box - RF Design Tab — Displays the RF group hierarchy within the design. Select an RF shape, node, or part to populate the property and preview fields. The RF Design tab enables you to define some of the RF group, RF shape, and net properties.

RF Control Dialog Box - RF Design Tab

To access: **Setup > RF > RF Control pane**

Description

Displays the RF group hierarchy within the design. Select an RF shape, node, or part to populate the property and preview fields. The RF Design tab enables you to define some of the RF group, RF shape, and net properties.

Figure 4-33. RF Control Dialog Box - RF Design

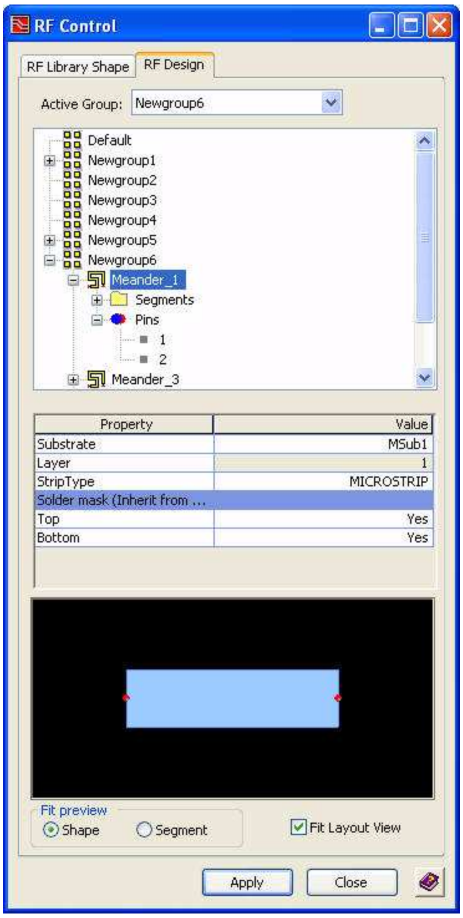


Table 4-7. RF Control Dialog Box - RF Design Tab Contents

Field	Description
Active Group	Defines the active RF group. Select an RF group from the dropdown list.
Treeview	Displays the RF group hierarchy. Select an RF group, RF shape, RF segment, or node to populate the **Property** table and preview area (Treeview Dropdown Menu Selections).

Table 4-7. RF Control Dialog Box - RF Design Tab Contents (cont.)

Field	Description
Property	Displays the selected RF group, RF shape, RF segment, or node properties. • Node Properties • RF Segment Properties • RF Shape Properties • RF Group Properties
Preview area	Displays the selected RF shape, RF segment, or node.
Fit preview section — Controls the preview area and layout tool display.	
Shape	Selected, displays an RF shape in the Preview area.
Segment	Selected, displays a segment in the Preview area.
Fit Layout View	Checked, zooms the selected item to fill the layout tool view.

Node Properties

Table 4-8 lists the node properties.

Table 4-8. RF Control Dialog Box Node Properties

Property Name	Description
Net	Defines the net. Connect different nets with an RF shape (Connecting Nodes and Pads with an RF Shape). **Note**: All nodes on a meander belong to the same net.
Node-to-node	Defines the entry angle for node-to-node connections, as defined in the RF Entry Rules Dialog Box. Set the entry angle with the **Option** dropdown list. • **Default** — Defines the entry angle as 180º. • **Fixed** — Defines the entry angle(s) as fixed. Type values in the **Values** field. Separate the entry angles with semicolons without spaces (For example: 135;180;225). • **Free** — Defines the entry angle as any angle.
Node-to-trace	Defines the entry angle for node-to-trace connections as defined in the RF Entry Rules Dialog Box. Set the entry angle with the **Option** dropdown list. • **Default** — Defines the entry angle as 180º. • **Fixed** — Defines the entry angles as fixed. Type values in the **Values** field. Separate the entry angles with semicolons without spaces (For example: 135;180;225). • **Free** — Defines the entry angle as any angle.

Table 4-8. RF Control Dialog Box Node Properties (cont.)

Property Name	Description

Connect-to-plane	Defines the type of node-to-plane connections. Use the **Value** dropdown list to set the connection state.
	• **Clear** — Prohibits a node to connect to a plane by default (No connect mode).
	• **Connect** — Enables a node to connect to a plane (Connect mode). Connect mode overrides the RF shape clearance setting (Clearance Editor Dialog Box). Connection occurs if:
	  • The plane and node belong to the same net.
	  • The RF shape edge is located inside or on the edge of the RF shape plane.

Expired	Defines the Design Rule Check (DRC) node reporting. An RF shape has a floating node with a parameter value of True, which enables DRC reporting (RF Expired Nodes and DRC RF Violations).
RFPort	Defines the node as an RFPort when True. Use the **Value** dropdown list to set the RFPort state. • **False** — Defines the node as Default (The node is not an RFPort). • **True** — Defines the entry angle as 180º. The default RFPort is visible when False. **Note**: The schematic capture tool uses a custom RF shape when at least one RFPort node is True. (Creating a Custom RF Shape).

RF Segment Properties

Table 4-9 lists the segment properties.

Table 4-9. RF Control Dialog Box RF Segment Properties

Property Name	Description
Substrate	Defines the substrate.
L	Defines the length of the segment.
W	Defines the width of the segment.
Solder-mask	Defines the solder-mask clearance for the selected segment. Refer to Soldermask Control.
Corner, Type	Defines the corner type.
Top/Bottom	Indicates if the soldermask settings are inherited from the group settings. • **Yes** —Soldermask setting are inherited. • **No** — Soldermask settings are not inherited.

RF Shape Properties

You create conductive or plane shapes in **Draw** mode and covert them to RF shapes (Creating a Custom RF Shape). Draw objects within RF elements that you modify are tagged as “user modified’ and stored as static shapes.

Note
You can assign different nets to nodes within the same RF shape.

Table 4-10 lists the RF shape properties.

Table 4-10. RF Control Dialog Box RF Shape Properties

Property Name	Description
Name	Displays the RF shape name. The name is the schematic reference designator (refdes) or a user-defined name. When you use the refdes name, the property is read-only.
W	Defines the RF shape width.
Layer	Defines the RF shape layer. This is the MSUB<*n*> in the RF Control Dialog Box.
Solder-mask	Defines the RF shape solder-mask (Soldermask Control).
L	Defines the total length of the RF shape. Modify the RF shape length with the RF Shape Parameters Dialog Box.
Strip type	Defines the RF shape as a Microstrip or Stripline.

RF Group Properties

RF group clearance rules have the same limitation as the RF shape level (RF Shape Properties). The RF group clearance rules are only set as a global rule on all RF shapes within the RF group (RF Group Clearance Rules Dialog Box).

The RF group properties populate the RF Control Dialog Box, RF Design tab - Property field when you select an RF group in the treeview area or in the layout tool.

RF shapes in an RF group inherit the RF group clearance values. An RF group level clearance rule is useful in setting the clearance rules on a large number of RF shapes.

Based on the RF group clearance rules, the layout tool displays a border around the RF group representing the defined clearances.

Treeview Dropdown Menu Selections

The treeview area menus (RF Control dialog box, RF Design tab) change when you right-click the following:

- RF Group — Table 4-11
- RF Shape — Table 4-12
- Varblock — Table 4-13

Table 4-11. RF Control Dialog Box, RF Group Dropdown Menu Selections

Dropdown Menu Selections	Description
Auto arrange	Enables the automatic arrangement of a placed RF group (Auto Arranger).
Generate netlist	Creates a netlist file and displays the contents in the default ASCII viewer (Generating Netlist).
Send netlist	Sends a netlist file to the RF simulator (Sending Netlist - Layout).
Generate layout data	Creates a momentum file and displays the contents in the default ASCII viewer (Generating Layout Data).
Send layout data	Sends a momentum file to the RF simulator (ADS Sending Layout Data, AWR/MWO Sending Layout Data).
Generate Tune/Opt Data	Creates a *.iff* file and displays the contents in the default ASCII viewer (ADS Generating Tune/Opt Data).
Send Tune/Opt Data	Sends a *.iff* file to the RF simulator (ADS Sending Tune/Opt Data).
Synchronize library	Synchronizes the layout tool library with the RF simulator library. Only those RF simulator library parts that have changed are updated in the layout tool library.
Add Var Block	Creates a variable block in a layout design (Adding a Varblock).
Paste Var Block	Pastes a copied Var block into an RF group.
Generate Library Symbol	Enables an RF shape to be placed in the library (Creating a Custom RF Shape).

Table 4-12. RF Control Dialog Box, RF Shape Dropdown Menu Selections

Dropdown Menu Selections	Description
Set Auto Place Seed	Selects the RF shape seed. The RF group or RF shape seed is the reference point for placement, rotation, and move functions.

Table 4-12. RF Control Dialog Box, RF Shape Dropdown Menu Selections

Dropdown Menu Selections	Description
Unset Auto Place Seed	Removes the active RF shape seed from the RF group. The RF group or RF shape seed is the reference point for placement, rotation, and move functions.

Table 4-13. RF Control Dialog Box, Var Block Dropdown Menu Selections

Dropdown Menu Selections	Description
Edit Var Block	Opens a Var Block (Creating Var Block Variables).
Delete Var Block	Removes the selected Var Block. Any parameters with variables, become regular values.

Related Topics

Editing Existing RF Shapes

Creating and Modifying Nodes

RF Entry Rules Dialog Box

To access:

- With the RF Toolkit () active, click **Entry rules** () (RF Toolkit Toolbar)
- **Setup > RF > Entry Rules**
- **RF > Entry Rules**

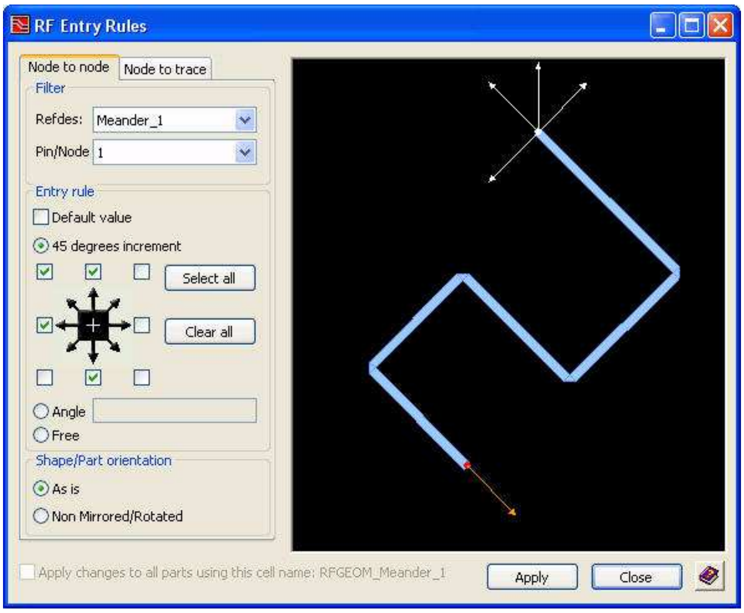
Description

Use the RF Entry Rules dialog box to define the node entry angles for node-to-node and node-to-trace nodes.

Changes made in the RF Entry Rules dialog box also displays in the RF Control Dialog Box, RF Design tab node properties area.

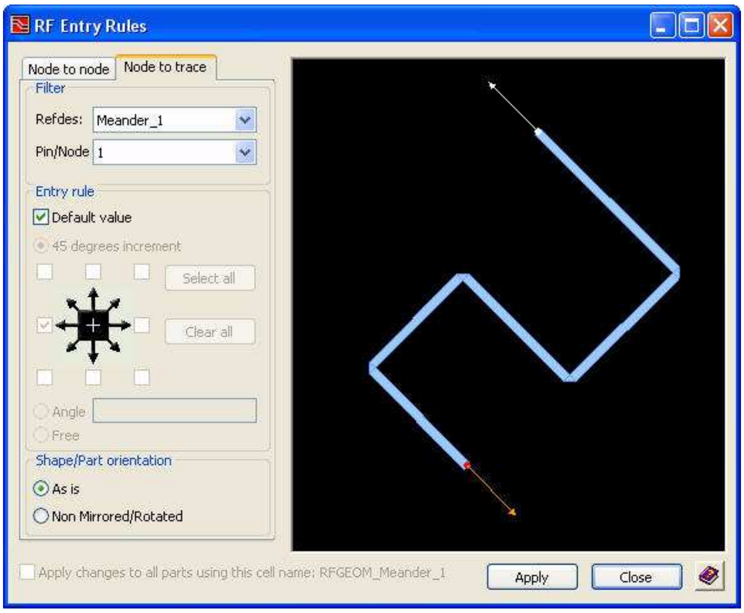
- Node entry angles ensure that traces, other RF shapes, and regular parts connect to a node at a specific angle.
- Node-to-node entry rules apply to auto-placement (Auto Arranger).
- The node-to-trace entry rule applies to all commands and functions that connect a node to a trace.

Figure 4-34. RF Entry Rules Dialog Box - Node to node tab



- Traces connecting to nodes use the centerline connection, which enables half the trace width to overlap the node (pin) (Automatic Meander Guide and Trim).
- RF shape nodes behave as pins do in **Route** mode and can have different entry rules.

Figure 4-35. RF Entry Rules Dialog Box - Node to trace Tab



The contents of the **Node to node** and **Node to trace** tabs are the same.

Table 4-14. RF Entry Rules Dialog Box Contents

Field	Description
Filter section — Defines the RF shape and node identification.	
Refdes	Defines the RF shape reference designator selection. Use the dropdown list to choose a different reference designator.
Pin/Node	Defines the RF shape node selection. Use the dropdown list to choose a different node.
Entry rule section— Defines the node entry angles. 90° 180° — ◈ — 0° ◈ Node 270°	
Default value	Checked, the entry angle is orthogonal to the RF shape edge (RF Shape Node Entry Angle). Unchecked, enables **45 degrees increment**, **Angle**, and **Free** options.
45 degrees increment	Checked, enables 45° selection.
Select all	Click, checks all 45° check boxes.

Table 4-14. RF Entry Rules Dialog Box Contents (cont.)

Clear all	Clears all 45° check boxes.
Angle	Selected, enables you to define entry angles. Type the entry angles separated by a semicolon without spaces (Example: 15;33;44;155 - entry angles are 15°, 33°, 44°, and 155°).
Free	Selected, disables angle checking and removes the entry angle indicator. The angle of entry is determined by **Plow** or the autorouter settings.
Shape/Part orientation section — Defines the RF shape orientation in the view area.	
As is	Selected, displays the layout tool view.
Non Mirrored/Rotated	Selected, displays the library view.
Apply changes to all parts using this cell name: *<cell_name>*	Checked, applies the changes to all parts with the same cell name, except locked or highlighted parts. Unchecked, applies the changes only to the selected node.

Related Topics

Selecting the Entry Angle During Meander Placement

Setting RF Shape Node Entry Angle

Creating and Modifying Nodes

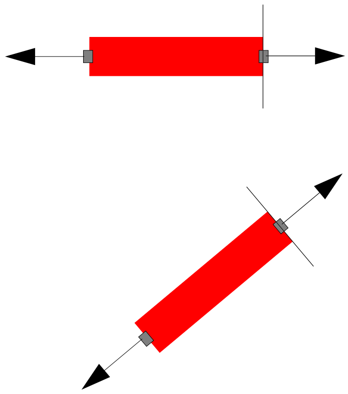
Creating and Modifying RF Shapes

RF Shape Node Entry Angle

When you add RF shapes to a design, the node entry angle is perpendicular to the RF shape edge by default. Rotating the RF shape does not change the node entry angle as shown in Figure 4-36.

The RF entry angle rule is a property on the node and not of the RF shape. After placing an RF shape, update the entry angle as needed (Setting RF Shape Node Entry Angle).

Figure 4-36. Default Entry Angle for RF Shapes



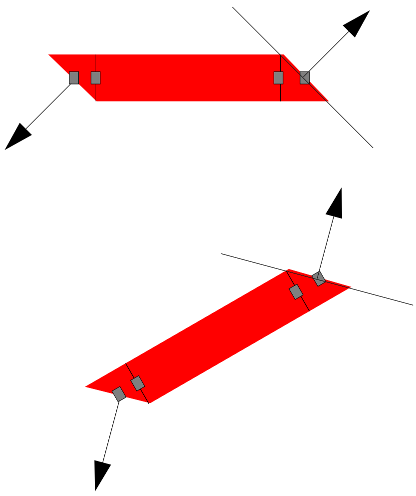
Entry angle is 0º

The default entry angle is perpendicular to the RF shape edge.

During rotation, the entry angle remains the same.

The node entry rules apply even if the RF shape is an arbitrary polygon. The entry angle is perpendicular to an arbitrary polygon RF shape edge as shown in Figure 4-37.

Figure 4-37. Default Entry Angle for Arbitrary Polygon



Entry angle is 45º

The default entry angle is perpendicular to the RF shape edge.

During rotation, the entry angle remains the same.

The entry angle is perpendicular to a microstrip curve as shown in Figure 4-38 or a custom RF shape as shown in Figure 4-39.

Figure 4-38. Default Entry Angle for Microstrip Curve (RF Shape/Meander)

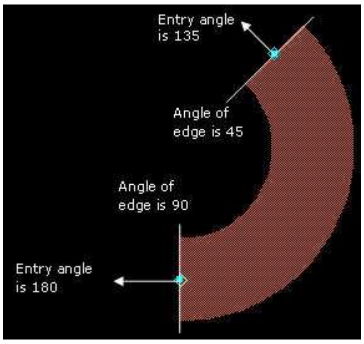
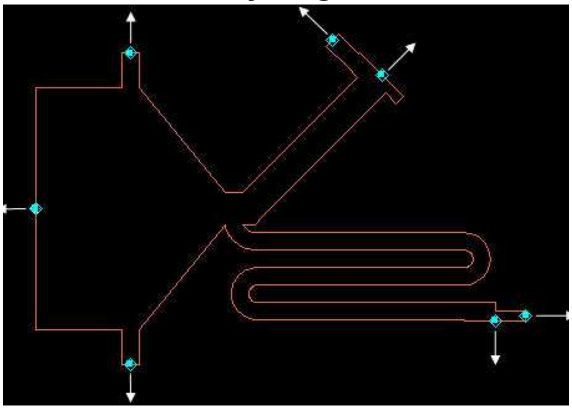
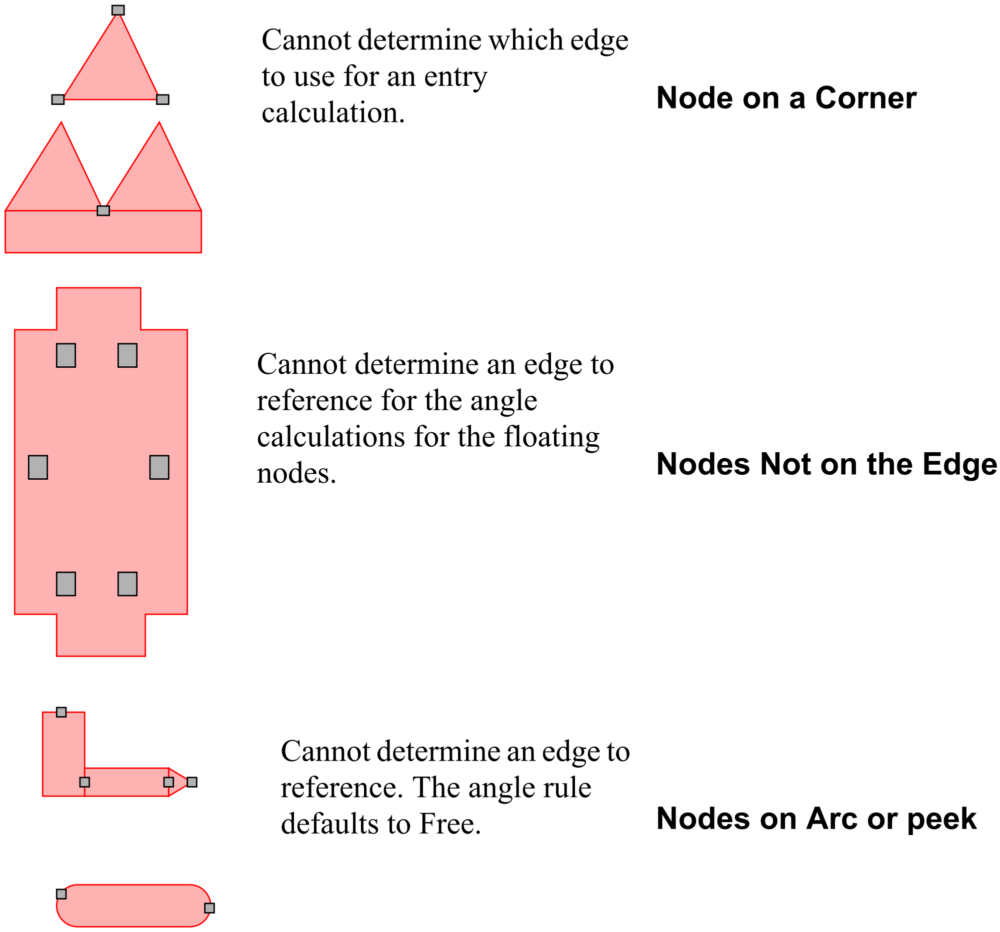


Figure 4-39. Normal Entry Angle for Custom RF Shape



If the custom RF shape edge cannot be used to determine the entry angle, the entry angle is set to Free (Figure 4-40).

Figure 4-40. Custom Shape Free Connections



Related Topics

Adding an Edge Node

Creating and Modifying Nodes

Editing Existing RF Shapes

Auto Arranger

Clearance Editor Dialog Box

To access:

- With the RF Toolkit ( ) active, click **Clearance rules** ( ) (RF Toolkit Toolbar).
- Choose **Setup** > **RF** > **Clearance Rules**.
- Select an RF segment or RF shape and choose **RF** > **Clearance Rules**.

Description

Define RF shape and segment clearances with the Clearance Editor dialog box.

Figure 4-41. Clearance Editor Dialog Box

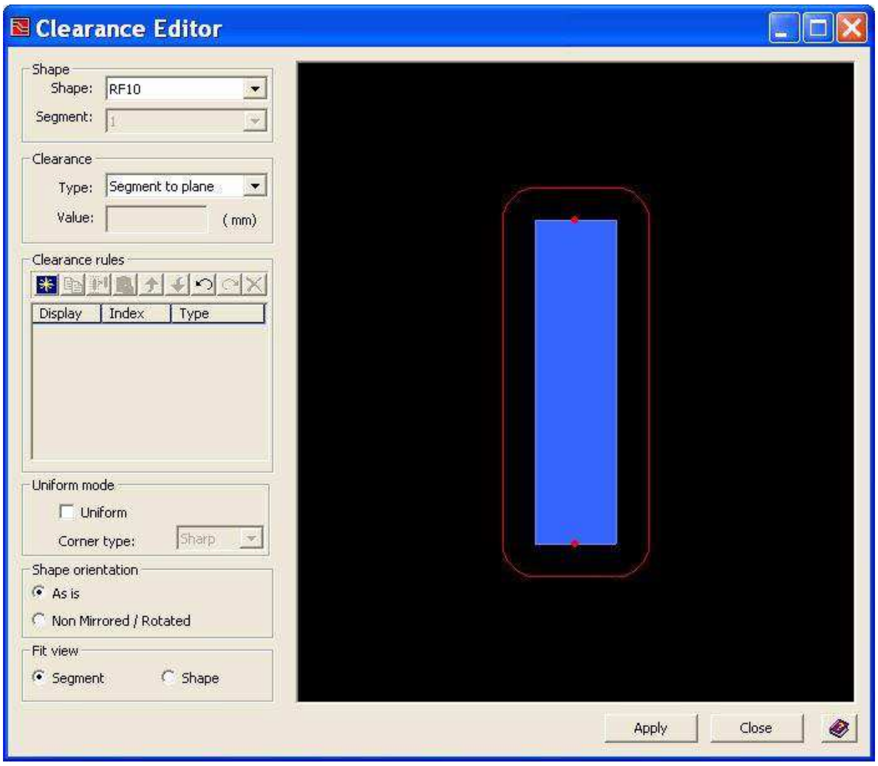


Table 4-15. Clearance Editor Dialog Box Contents

Field	Element Description
Shape	Defines the selected RF shape and segment. • **Shape** — Displays the name of the selected RF shape or segment. Select RF shapes from the dropdown list (RF Shape Clearance). • **Segment** — Displays the RF segment number of the selected RF shape. Select the segment from the dropdown list (New Segment Clearance Rule).

Table 4-15. Clearance Editor Dialog Box Contents (cont.)

Field	Element Description
Clearance section — Defines the RF shape or segment clearance.	
Type	Defines the clearance type. The following types are available from the dropdown list: • **All** — Default value, which applies to all of the segment settings except for the Segment to mask clearance. • **Segment to plane** — Defines the segment edge to plane clearance. • **Segment to trace** — Defines the segment edge to trace clearance. • **Segment to via** — Defines the segment edge to via clearance. • **Segment to pad** — Defines the segment edge to pad clearance. • **Segment to mask** — Defines the segment edge to mask clearance (Soldermask Control).
Value	Defines the clearance measurement value for the clearance type.
Clearance rules section — Defines the RF segment layer clearance rules (Setting Clearances, Copying, and Creating Adjacent Layer Clearances).	
New	Click, adds a new clearance rule for the segment as defined by the Clearance field Type selection (New Segment Clearance Rule). New table columns are: • **Display** — Checked, RF shape or segment clearances display in the dialog box viewer (RF Segment Clearance). • **Index** — Defines the clearance layer. The segment layer has an index value of 0. Negative numbers defines the layers above the segment layer. Positive numbers defines the layers below the segment layer. • **Type** — Defines the layer type as Signal, Plane and so on. Non-existent layers are Unknown.
Copy	Click, copies a Clearance field Type clearance to the clipboard (Related Topics).
Copy To Other Shape	Click, copies all the clearance rules to the clipboard. **Note**: You cannot copy a tapered clearance to an RF corner shape.
Paste	Click, applies the clipboard clearances to the same kind of RF segment selection when the clipboard contains a Copy to Other Shape. Click, applies the clipboard clearances to a clearance type when the clipboard contains a Copy.
Increase Layer Index	Click, increments the Index value for layers below the RF shape. This enables a per layer clearance setting (Layer-to-Layer Clearance Rules). Use the layer index to define a clearance setting on layers below the RF shape.
Decrease Layer Index	Click, decrements the Index value for layers above the RF shape. This enables a per layer clearance setting (Layer-to-Layer Clearance Rules). Use the layer index to define a clearance setting on layers above the RF shape.
Undo	Click, restores the last deleted clearance rule.

Table 4-15. Clearance Editor Dialog Box Contents (cont.)

Field	Element Description
Redo	Click, reverses the Undo.
Delete	Click, Deletes the selected clearance rule.
Uniform	Checked, displays all the RF shape or segment clearances in a message window (Figure 4-47). The largest clearance value becomes the uniform clearance for that clearance type (Uniform Mode). The clearance around the RF shape become uniform. Unchecked, enables per-side segment clearances. Each side of the segment can have a different clearance.
Corner type	Enabled with Uniform checked, defines the clearance corner as sharp (angle) or rounded (radius).
Shape orientation — Defines the orientation of the selected shape in the dialog box viewer.	
As is	Selected, displays the selection in the dialog box viewer as it appears in the layout tool.
Non Mirrored / Rotated	Selected, displays the library orientation in the dialog box viewer.
Fit view — Defines the view in the dialog box viewer.	
Segment	Selected, displays the selected segment in the dialog box viewer.
Shape	Selected, displays the selected RF shape in the dialog box viewer.

New Segment Clearance Rule

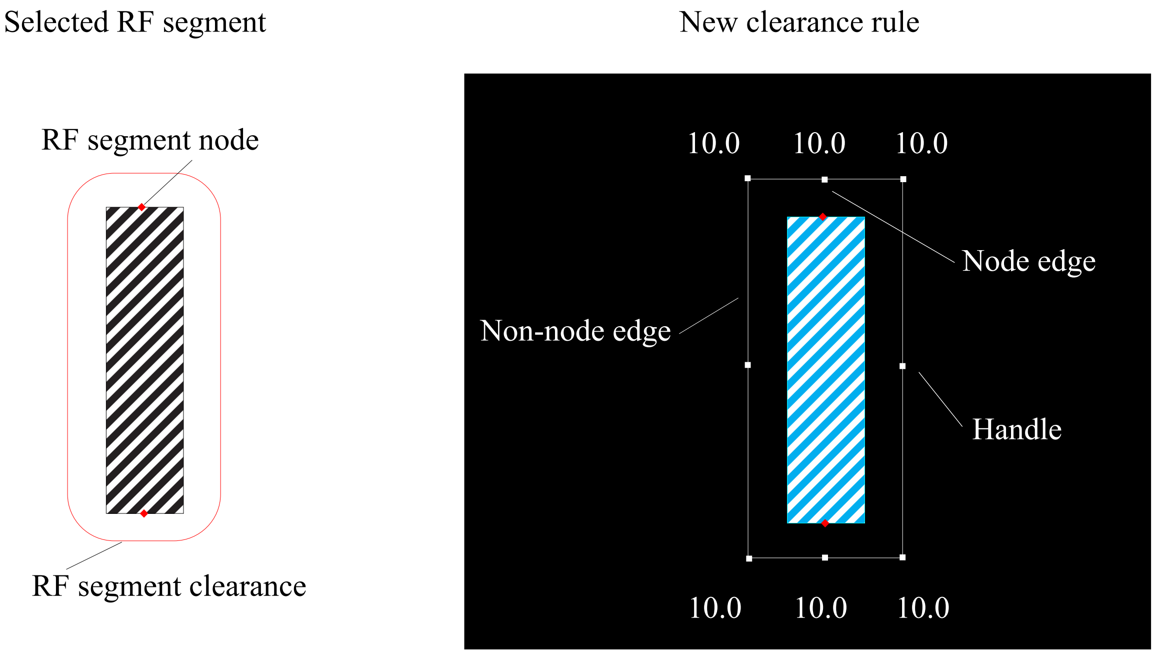
The selected segment appears in the Clearance Editor Dialog Box with a red segment clearance (Figure 4-42). Selecting New () displays the default clearance as defined by the white border around the RF segment. The white border, values, and handles (square blocks) indicate the default segment clearance. Clearances for an RF segment can be per-side or uniform. A per-side clearance means each side can have a different clearance value. A uniform clearance means all sides have the same clearance value.

Non-node clearance edges have three handles which move the border perpendicular to the RF segment edge. Node clearance edges have one handle, which moves the border perpendicular to the RF segment edge.

Select the center handle to change the edge clearance uniformly. The white clearance edge turns red, which enables you to move the clearance edge or enter a value in the Clearance Editor, Clearance field, Value.

Select a corner handle to create a tapered clearance. The white clearance edge turns red, which enables you to move the corner handle or enter a clearance in the Clearance Editor, Clearance field, Value.

Figure 4-42. RF Segment Clearance Handles



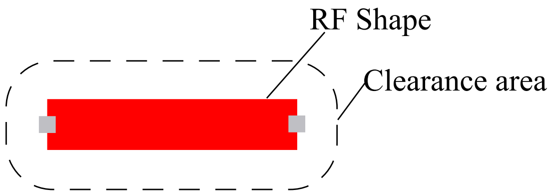
RF Shape Clearance

When an RF Shape contains more than one segment, you can change the clearances of each segment of the RF shape. The RF shape supports only a single clearance value. Each segment within the RF shape can support a different value for each side of the segment (RF Segment Clearance).

RF Segment Clearance

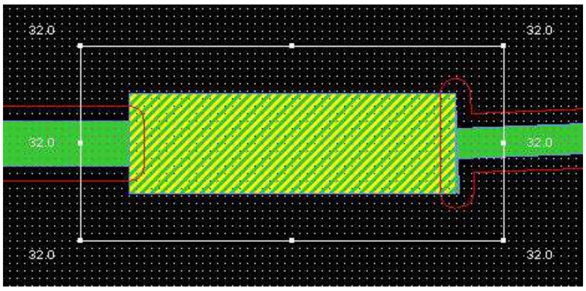
Figure 4-43 shows an RF shape with a uniform clearance because the clearance has radius corners. Each segment and RF shape has a default clearance. An RF shape or segment with a uniform clearances can have sharp (angle) or Rounded (radius) corners. An RF shape or segment with a non-uniform clearance can only have sharp (angle) corners.

Figure 4-43. RF Segment Clearance



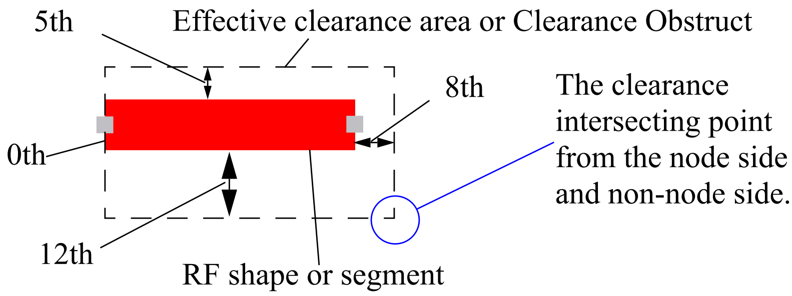
Each segment of an RF shape can have different set of clearances (Figure 4-44).

Figure 4-44. Clearance - Different Segment Values



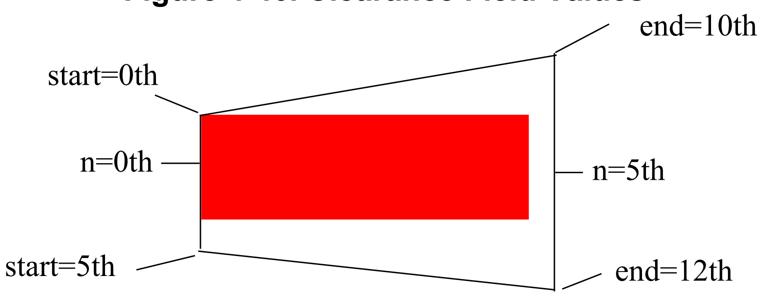
Each side of an RF shape or segment can be set to a different clearance (Figure 4-45).

Figure 4-45. Clearance - Different RF Segment Rules



A tapered RF segment clearance value applies to the segment clearance and not the RF shape rule (refer to RF Shape Clearance Restrictions and Exceptions). Figure 4-46 shows an RF segment with tapered clearance. The different clearance values from the node and non-node sides of the segment, determine the effective clearance.

Figure 4-46. Clearance Field Values



Note

Obstructs are created for all RF shapes. One obstruct is created per clearance value and if the clearance values are the same, only one clearance displays.

Related Topics

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using RF Groups

Editing Existing RF Shapes

Creating and Modifying RF Shapes

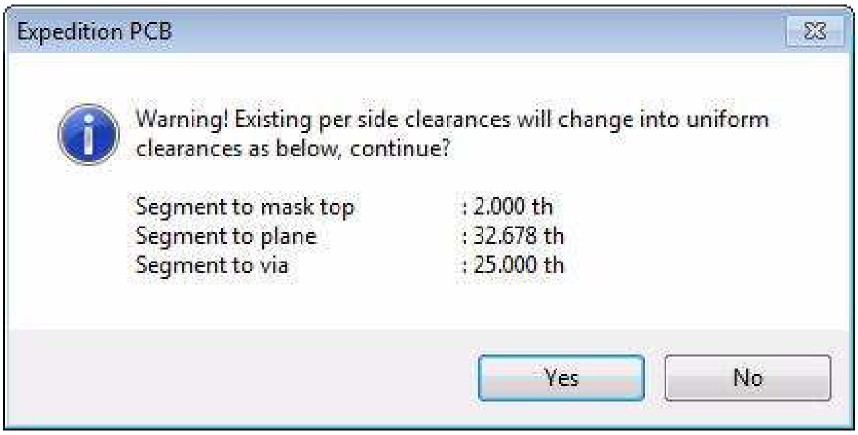
Uniform Mode

The Clearance Editor Dialog Box Uniform mode field enables conversion from a per-side clearance to a uniform clearance.

For example:

A segment has different per-side clearances and when you select Uniform in the Clearance Editor Dialog Box and click **Apply** a message window displays as shown in Figure 4-47.

Figure 4-47. Uniform Message Window



If you click **Yes**, the largest per-side clearance of the segment type becomes the uniform clearance. In this example, a uniform clearance becomes 32.678th.

If you do not specify a specific corner type in the Clearance Editor, Clearance field, Value, the RF group corner prevails. The RF Group Clearance Rules Dialog Box defines the segment default corner.

Note

A meander with multiple segments, can have a per-side and uniform clearances for each segment.

Related Topics

Editing Existing RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Auto Arranger

DRC RF Violations

Soldermask Control

The Soldermask clearance can be set at the RF group, RF shape, or RF segment level.

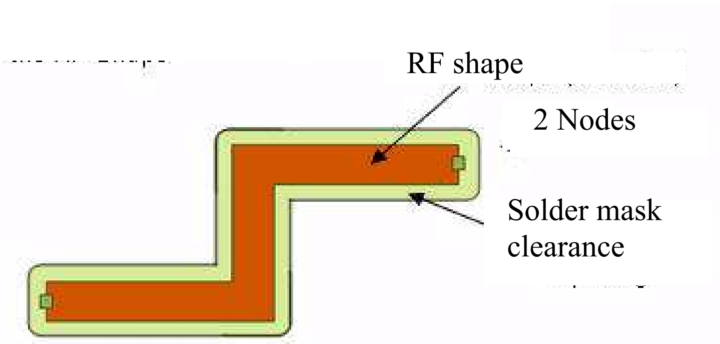
The soldermask property appears in the RF Control Dialog Box, RF Design tab Properties field with an RF group selected. Change the soldermask value with the Setting Clearances, Copying, and Creating Adjacent Layer Clearances procedures.

You can set the soldermask value at the RF group level with the RF Group Clearance Rules Dialog Box (Top Solder, Bottom Solder). You can set the soldermask value at the segment level with the Clearance Editor Dialog Box, clearance type Segment to mask.

The soldermask:

- can be larger or smaller than the width of the RF segment.
- automatically adjusts to the RF shape segments width.
- follows the contour of the RF shape as shown in Figure 4-48.
- is not affected by **Push** or **Mirror** operations.

Figure 4-48. Soldermask Opening



Note
If the RF shape is pushed to an inner layer of the design, you must manually turn off the soldermask clearance.

Note
Soldermask clearances on an RF group, RF shape, or RF segment, move with the RF shape.

The RF shape soldermask clearances have the following:

- A per-side control for the following RF shape types (RF Shapes):

    - Microstrip corner
    - Microstrip line
    - Microstrip taper
- A tapered soldermask for the following RF shape types (RF Shapes):
    - Microstrip line
    - Microstrip taper

Meanders can contain multiple microstrip RF shapes, while supporting the same per-side and tapered soldermask clearances.

Related Topics

Clearance Editor Dialog Box

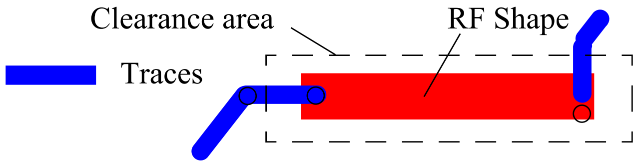
RF Layout

RF Shape Clearance Checking

Trace Connection Clearance

A trace must clear an RF shape clearance area before it can change direction. This applies to all traces that connect to RF shape nodes (RF Shape Node Types). Violations occur when a trace re-enters the RF shape clearance area. A trace can only enter an RF shape clearance area once.

Figure 4-49. RF Shape/Segment to Trace



Related Topics

Creating a Meander

Adding a Stub

Adding a Segment

DRC Node and Pad Entry Rules

Connecting RF Shapes by Moving

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Use the following to set and define RF shape clearances:

- Setting RF Segment Clearance
- Copying
- Copying to Other Shape
- Creating Adjacent Layer Clearances

Related Topics

Creating and Modifying RF Shapes

Editing Existing RF Shapes

Clearance Editor Dialog Box

Setting RF Segment Clearance

Use the following to set the RF segment clearance:

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF segment in the Clearance Editor Dialog Box.
3. Select clearance type.
4. Click **New** ( ).
5. Move the clearance handle, set the Value field, or check **Uniform** to change clearances (New Segment Clearance Rule).
6. Click **Apply**.

Related Topics

Creating and Modifying RF Shapes

Editing Existing RF Shapes

Clearance Editor Dialog Box

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Copying

Use the following to copy the current clearance of an RF shape or segment:

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF segment in the Clearance Editor Dialog Box.

3. Select clearance type.
4. Click **New** ( ).
5. Move a clearance handle, set the Value field, or check Uniform to define a clearance (New Segment Clearance Rule).

Note
Copy is active when an RF shape or segment has different clearance type clearances.

6. Click **Copy** ( ). The Copy clearance to dialog box displays with the current clearance type grayed.
7. Check clearances as needed.
8. Click **OK**
9. Click **Apply**

Related Topics

Creating and Modifying RF Shapes

Editing Existing RF Shapes

Clearance Editor Dialog Box

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Copying to Other Shape

Use the following to copy the RF shape clearance and apply the clearance to a similar RF shape.

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF segment in the Clearance Editor Dialog Box.
3. Select clearance type.
4. Move a clearance handle, set the Value field, or check Uniform to define a clearance (New Segment Clearance Rule).
5. Click **Copy To Other Shape** ( ).
6. Select a similar RF shape or segment.
7. Click **Paste** ( ).
8. Click **Apply**.

Related Topics

Creating and Modifying RF Shapes

Editing Existing RF Shapes

Clearance Editor Dialog Box

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating Adjacent Layer Clearances

Use the following to create adjacent layer clearances for the selected RF shape:

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF segment in the Clearance Editor Dialog Box.
3. Select clearance type.
4. Click **New** ( ) (New Segment Clearance Rule).
5. Move a clearance handle, set the Value field, or check Uniform to define a clearance.
6. Increase or Decrease Layer Index value to define a layer with the ( ) arrows (Layer-to-Layer Clearance Rules).
7. Click **Apply**.

Related Topics

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

RF Shape Clearance Checking

RF Shape Layer Change - Clearances

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Group/Ungroup Dialog Box

To access:

- With the RF Toolkit ( ) active, click **Group/Ungroup** ( ) (RF Toolkit Toolbar).
- Choose **Setup > RF > Group/Ungroup**.
- Click action key **11 Group/Ungroup** with an RF shape selected.
- Choose **RF > Group/Ungroup** with an RF shape selected.
- Choose **RF > Group/Ungroup**.

Description

Create RF groups containing RF shapes and non-RF shapes with the Group/Ungroup dialog box.

Create RF groups by function and move members between RF groups as needed (Moving Members Between RF Groups). RF groups with subgroups are sent to the RF simulator for simulation (RF Group Elements Exported to RF Simulator and RF Simulator).

Figure 4-50. Group/Ungroup Dialog Box Selection

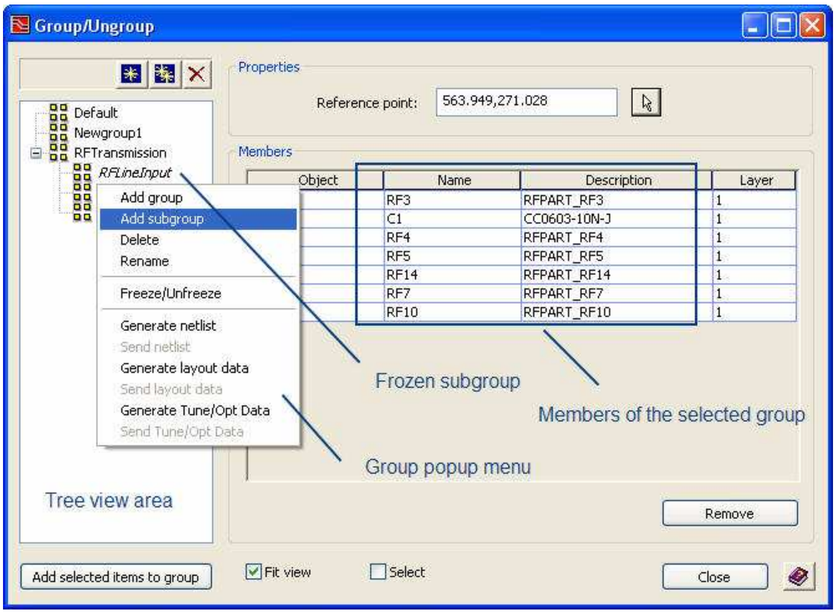


Table 4-16. Group/Ungroup Dialog Box Contents

Field	Description
Add Group	Adds an RF group at the selected RF group level (Adding an RF Group or RF Subgroup). The name of a frozen RF group is italicized (Freeze/Unfreeze an RF Group). Choose an RF group to view (RF Group Menu Selections).
Add Subgroup	Adds an RF subgroup under the selected RF group level (Adding an RF Group or RF Subgroup). The name of a frozen RF group is italicized (Freeze/Unfreeze an RF Group). Choose an RF group to view (RF Group Menu Selections).
Delete	Deletes the selected RF group and all the members are placed in the Default group.
Treeview area	Displays the RF group structure. Click the plus-symbol to expand and access subgroups. Choose RF groups to access the RF Group Menu Selections.
Properties — Defines the reference point for the object selected.	
Reference point	Displays the RF Group or Pick selection location. The RF group's default reference point is the board origin.
Pick reference	Click, enables you to define one of the following reference points: • **Location coordinates** — Click **Pick reference** ( ) and click a point in the work area. The location coordinates display. • **An RF group object** — Click **Pick reference** ( ) and click an RF group object. The location coordinates display. **Note**: The object is referenced, not the coordinates of the object. The reference point of the RF group moves with the RF object.
Members	

Table 4-16. Group/Ungroup Dialog Box Contents (cont.)

Field	Description
Table columns	Displays the selected RF group members information. **Object** — Displays the object type (Objects in RF Groups - Special Conditions). • **Conductive** — Conductive shape • **Group** — RF group within an RF group (subgroups) • **Parts** — Layout parts • **Draw** — Plane, Route obstructs • **Trace** — Traces • **Vias** — RF Vias **Name** — Displays the objects name, subgroup name, or reference designator. Non-named objects have the following displays: • **Trace** — Net name • **Vias** — Padstack name • **Draw** — Type of draw object, which are route obstructs, plane obstructs and so on. **Description** — Displays a description of the part or reference location. **Layer** — Displays the part layer or via span.
Remove	Click, removes selected members from the RF group (RF Group Contents) and moves them to the Default RF group.
Add selected items to group	Click, adds selected design objects to the selected RF group (Adding Members to an RF Group).
Fit view	Checked, magnifies the selected item to fill the layout tool view.
Select	Checked, enables crossprobing, select a member and it is highlighted in the layout tool.

Related Topics

Using RF Groups

Editing Existing RF Shapes

RF Group Contents

An RF group can contain a parts, fills, traces, vias, draw objects, as well as RF shapes. The RF group objects are called members. Each RF group has a unique name, which displays in the Group/Ungroup dialog box, treeview area (Figure 4-50).

An RF group can contain the following objects:

- Conductive Shape
- Group
- Netline
- Pad Obstruct
- Part
- Pin
- Plane Obstruct
- Plane Shape
- RF Node
- RF Segment
- RF Shape
- Route Obstruct
- Trace
- Via

RF groups are sent to the RF simulator with the dynamic-link (RF Simulator).

Note

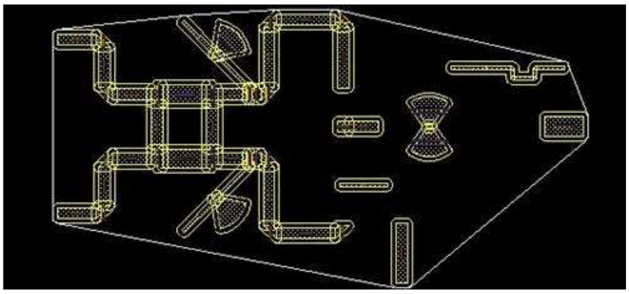
Traces are only sent to the RF simulator if they are directly or indirectly connected to an RF shape or meander (RF Group Elements Exported to RF Simulator).

Figure 4-51 shows a top level RF group border (white line), which encloses an RF group (RF Group Clearance Rules Dialog Box). RF subgroups do not have a group border.

Note

To select an RF group you check **Group** in the Selection Filter Dialog Box, Object tab.

Figure 4-51. RF Group Border



Note

RF groups created with the Group/Ungroup dialog box are only accessible through the RF Toolkit. The Group/Ungroup dialog box defines the group as an RF Group.

The Default RF group members change when you:

- Add an RF part with no additional RF group.
- Delete members from an RF group.
- Delete an RF group that has members.

Related Topics

Using RF Groups
Moving Members Between RF Groups
RF Group Properties
Group/Ungroup Dialog Box
Editing Existing RF Shapes
RF Group Menu Selections
Selection Filter Dialog BoxGroup/Ungroup Dialog Box

RF Group Menu Selections

The following popup menu selections are available when you choose an RF group in the Group/Ungroup Dialog Box treeview area.

- **Add group** — Adds an RF group at the selected RF group level (Adding an RF Group or RF Subgroup).
- **Add subgroup** — Adds an RF subgroup under the selected RF group (Adding an RF Group or RF Subgroup).
- **Delete** — Deletes the selected RF group and moves them to the Default group.
- **Rename** — Enables renaming of the RF group.
- **Freeze/Unfreeze** — Enables Freezing and unfreezing of an RF group (Freeze/Unfreeze an RF Group).

- **Generate netlist** — Displays the RF group ASCII netlist file (Generating Netlist).
- **Send netlist** — Sends the RF group netlist file to the RF simulator (Sending Netlist - Layout).
- **Generate layout data** — Displays the RF group ASCII layout data file (Generating Layout Data).
- **Send layout data** — Sends the RF group layout data to the RF simulator (Sending Layout Data).
- **Generate Tune/Opt Data** — Creates a Tune/Opt data file (*.iff*) in a location shown by the message window (Generating Tune/Opt Data).
- **Send Tune/Opt Data** — Sends only RF elements (model) with parameters and breaks down meanders into standard RF simulator models and parts (Sending Tune/Opt Data).

Related Topics

Group/Ungroup Dialog Box
Freeze/Unfreeze an RF Group

Freeze/Unfreeze an RF Group

You can freeze an RF group when all the parts are placed in the layout tool. A frozen RF group and its subgroups cannot be deleted or manipulated. You can move, rotate, or mirror a frozen RF group.

Use the RF group dropdown list **Freeze/Unfreeze** to freeze or unfreeze an RF group. A frozen RF group name displays in *italics* in the Group/Ungroup Dialog Box and in the RF Control Dialog Box - RF Design tab treeview areas.

Related Topics

Moving RF Groups or Objects in RF Groups
Objects in RF Groups - Special Conditions

Objects in RF Groups - Special Conditions

The following RF group parts are treated in a special way when they are RF group members:

- Traces and Vias
- Parts

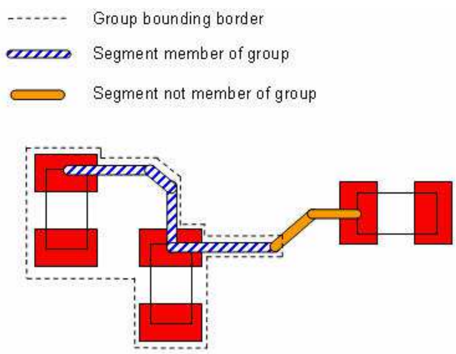
Traces and Vias

Traces and vias are locked when they are members of an RF group. The trace or segments of a trace that are members, move with the RF group. A trace that is a member of the RF group cannot be manipulated, pushed, or shoved. Moving an RF group that contains a non-RF group trace, causes the trace to disconnect from the RF group. Moving an RF group with an attached non-RF group trace, causes the trace to disconnect from the RF group.

The trace rules apply to regular vias (vias created with the Padstack Editor). A via can only be independently moved within an RF group when it is not connected to traces (stitched vias Vias in RF Designs). Vias can be indirectly moved as described in Parts.

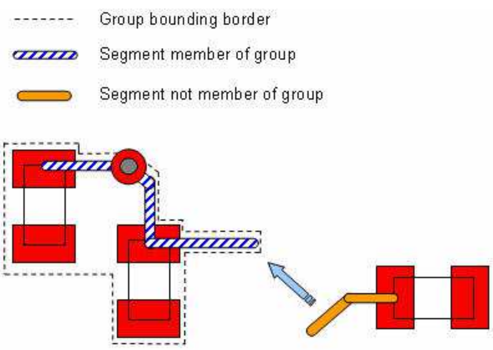
Figure 4-52 shows a trace (blue) that is part of an RF group, which prohibits trace manipulation. Remove the trace from the RF group for trace manipulation (Group/Ungroup Dialog Box **Delete** button).

Figure 4-52. Locked Traces in Groups



If you move an RF group in Figure 4-52 and the orange trace segment is not part of the RF group, that trace segment does not move with the RF group (Figure 4-53). Use **Plow** to reconnect the broken trace without changing segment membership.

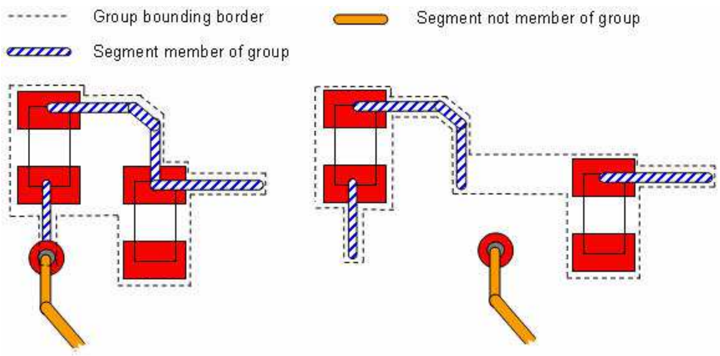
Figure 4-53. Trace Split After Group Move



Parts

Moving an RF group part member that is connected to a non-RF group via disconnects as shown in Figure 4-54. Move traces and non-stitched vias within an RF group only when you invoke Circuit Move & Copy (CMC) with an active RF toolkit. Move a fanout part, via, and trace by selecting CMC prior to layout selection. When you move a part, all the traces and vias connected to the pin (pad) or a branch move with the part. A branch is a trace T-junction, where the trace segment is not a member of the RF group, via, or another pin (pad).

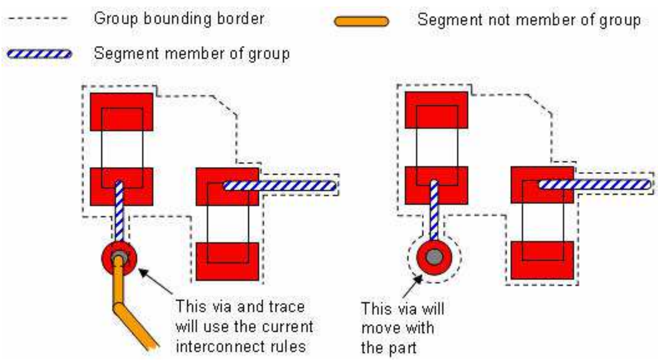
Figure 4-54. Move a Part in an RF Group



When you move a part, traces connected to the part or shape are treated as locked and you cannot **Push** or **Shove** other objects.

If a via is only connected by a single segment (fanout), the via moves with the part (Figure 4-55).

Figure 4-55. Move Part with Fanout Via



Related Topics

Using RF Groups	Editing Existing RF Shapes
Creating a Meander	Using RF Groups
Moving RF Groups or Objects in RF Groups	Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using RF Groups

Use the following to create and manipulate RF groups:

- Adding an RF Group or RF Subgroup
- Moving Members Between RF Groups
- Adding Members to an RF Group
- Moving RF Groups or Objects in RF Groups
- Placing RF Groups with Auto Arranger

Related Topics

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using Variables

Group/Ungroup Dialog Box

Adding an RF Group or RF Subgroup

Add RF groups or RF subgroups to reduce the member count within an RF group or to group by function (RF Simulator).

Use the following procedure to add an RF group or RF subgroup to the Group/Ungroup dialog box treeview area.

Prerequisites

- The design contains RF groups.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF group or RF subgroup in the treeview area (Meander Properties Dialog Box).
3. Click **Add group** () or **Add subgroup** ().
4. Select the new RF group name and choose **Rename**.
5. Type in an RF group name.

Results

A new RF group displays in the Group/Ungroup dialog box treeview area.

Related Topics

Using RF Groups

Using Variables

Selection Filter Dialog Box

Creating and Modifying RF Shapes

Replacement and Repair

Group/Ungroup Dialog Box

Moving Members Between RF Groups

Move members to different RF groups to maintain function, reduce member count, or for better RF Simulator results.

Prerequisites

- The design contains RF groups.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Group/Ungroup** ( ).
3. Select the RF group having the member(s) to be moved.
4. Select member(s) to be moved. Use **Ctrl** or **Shift** to select more than one item.
5. Hold down the **Ctrl** key and drag-and-drop the member(s) to the new RF group in the treeview area.

Results

Selected members belong to a different RF group.

Related Topics

Using RF Groups
Using Variables
Selection Filter Dialog Box
Creating and Modifying RF Shapes
Replacement and Repair
Group/Ungroup Dialog Box

Adding Members to an RF Group

Add additional parts, components, and so on to an existing RF group.

Prerequisites

- The design contains RF groups.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Group/Ungroup** ().
3. Select the RF group.
4. Select objects in layout that you want to be members of the RF group.
5. Click **Add Selected items to group**.

Results

The selected objects are members of the RF group.

Related Topics

Using RF Groups

Using Variables

Selection Filter Dialog Box

Creating and Modifying RF Shapes

Replacement and Repair

Group/Ungroup Dialog Box

Moving RF Groups or Objects in RF Groups

Move an RF group or objects in an RF group before RF simulation. Moving an object within an RF group does not affect its membership within the RF group. A member of an RF group remains a member after a move, even if it is not physically connected to any object within an RF group.

An RF group or individual objects within an RF group can be moved. The RF group border may change depending on member placement (RF Group Contents).

Prerequisites

- The design contains RF groups.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Group/Ungroup** ().
3. Select the RF group containing members (Adding Members to an RF Group).
4. Select the RF group or object (Selection Filter Dialog Box - Object tab, Group checked).
5. Select Action Key **2 Move**.

   The selected RF group or object attaches to the cursor. Change the Changing Snap Points by pressing the **Tab** key.
6. Click to place the RF group or object.

Results

The RF group or object is in a new location.

Related Topics

Using RF Groups
Using Variables
Auto Arranger
Creating and Modifying RF Shapes
Replacement and Repair
Group/Ungroup Dialog Box

Placing RF Groups with Auto Arranger

RF groups in schematic capture are forward-annotated and placed in the layout tool.

Prerequisites

- RF groups in schematic capture must be forward-annotated to layout.

Procedure

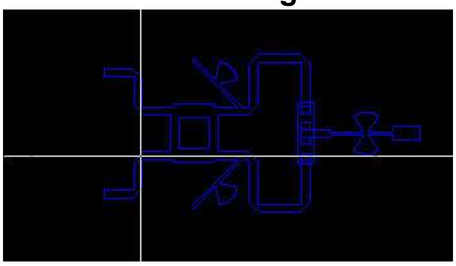
1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Choose **Place > Place Parts and Cells** and select RF Group from Criterion.
3. Select an RF group(s) and move them to the **Active** field ( ).
4. Click the **Method** dropdown arrow and choose Automatic (Auto Arranger).
5. Expand the RF group and select the RF group seed.

   The seed is the stationary RF group reference point during placement.
6. Click **Apply**.

   A ghost image circuit attaches to the cursor at the seed point (Figure 4-56). Change the seed during placement with the **Tab** key.
7. Click to place the RF group.

   If you selected more than one RF group, the next RF group automatically attaches to the cursor.

Figure 4-56. Auto-Arranger Ghost Image



Related Topics

Using RF Groups

Using Variables

Changing Snap Points

Creating and Modifying RF Shapes

Replacement and Repair

Group/Ungroup Dialog Box

RF Group Clearance Rules Dialog Box

To access:

- Choose **Setup > RF > Group Clearance Rules**
- With the RF Toolkit () active, choose **RF > Group Clearance Rules**

Description

Use this dialog box to set the RF group level clearances for an RF shape to trace, pad, via, plane, and top and bottom solder mask.

The RF group clearance rule is an RF group level rule defining the minimum distance from the RF shape edge to a trace, pad, via, plane, and solder mask (Figure 4-57).

Figure 4-57. RF Group Clearance Rules Dialog Box

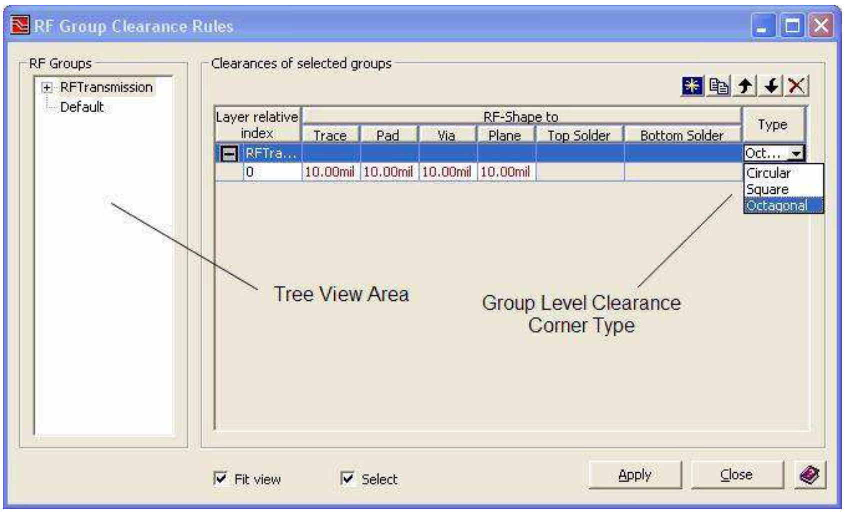
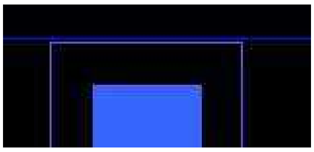
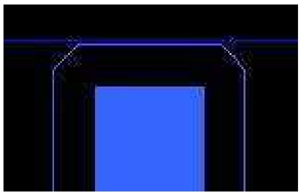


Table 4-17. RF Group Clearance Rules Dialog Box Contents

Field	Description
RF Groups	Displays all RF groups in the layout tool within the treeview area. The treeview area displays the RF group hierarchy.
Clearances of selected groups section — Displays the selected RF group level clearances (RF Shape Clearance Checking).	
New	Click, adds a new relative layer index rule.
Copy	Click, copies the currently selected rule and creates a new relative layer index rule.
Increase Layer Index	Click, increments the Index value so you can set clearance rules on adjacent layers below the RF group. The relative layer of the RF group has an index value of 0.

Table 4-17. RF Group Clearance Rules Dialog Box Contents (cont.)

Field	Description
Decrease Layer Index	Click, decrements the Index value so you can set clearance rules on adjacent layers above the RF group. The relative layer of the RF group has an index value of 0.
Delete	Click, deletes the selected layer index rule.
Layer relative index	Defines the clearance rule layer. The RF group relative layer has an index value of 0. Positive index values apply to layers below the RF group relative layer. Negative index values apply to layers above the RF group relative layer.
RF-Shape to	Displays the minimum clearance values from the RF shape to the following elements: • **Trace** — Defines the RF shape to trace distance (Trace, Via, and Pad Clearances and Connections). • **Pad** — Defines the RF shape to pad distance. • **Via** — Defines the RF shape to via distance (Vias in RF Designs). • **Plane** — Defines the RF shape to plane distance (RF Shape Clearance Checking). • **Top Solder/Bottom Solder** — Defines the RF shape solder-mask clearance at the RF group level (Soldermask Control).
Type	Defines the RF group corner type at the RF group level. The Type value controls the RFGroupClearance value in the RF Group Properties dialog box. Click the RF group level Type dropdown arrow and choose one of the following RF group clearance corners: Circular Square Octagonal   
Fit view	Checked, magnifies the selected item to fill the layout tool view.
Select	Checked, selects the layout item.

Related Topics

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using RF Groups

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Editing Existing RF Shapes

RF Shape Clearance Checking

Selection Filter Dialog Box

Selection Filter Dialog Box

To access: With the RF Toolkit () active, click **Selection filter** () (RF Toolkit Toolbar).

Description

The Selection Filter dialog box defines the selectable objects in the layout tool.

The following selections are applicable with RF designs:

- **Netlines**
- **Pins**
- **Group**
- **RF Nodes**
- **RF Segments**
- **RF Shapes**

Related Topics

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating and Modifying a Meander

Creating and Modifying Nodes

Using RF Groups

Editing Existing RF Shapes

Vias in RF Designs

Vias in RF designs are of the following types:

RF via — Carefully modeled via that is a part including a reference designator of the circuit simulation. Place the RF via part in the design with the schematic capture tool. RF vias have a schematic representation and have simulation models available within the RF Simulator.

An RF via is like any other via with respect to object connections. Use the RF via to connect traces, planes, or conductive shapes. The RF via has RF via entry and exit pad shapes. The RF via has only two connections, the entry and exit layer.

Stitch via — Regular vias (standard), created with the Padstack Editor, are associated with a net name. Use stitch vias to create ground shields, to ground an RF shape, or to connect RF shapes on different layers. Stitch vias are placed in an exact position as required by the RF circuit. They are not moved or removed by any optimization method unless explicitly requested.

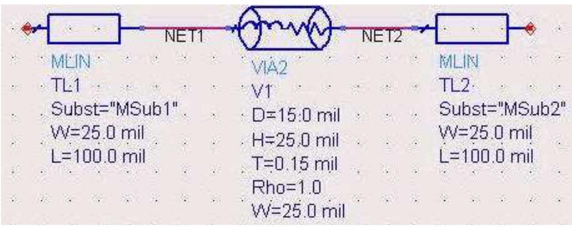
The following via topics are available:

- Standard Vias and RF Nodes
- Place Via Dialog Box
- Continuing a Meander on a Different Layer

Describe an RF Via as a pad stack part with an entry pad, exit pad, and drill hole. The RF via drill hole only exists for manufacturing purposes. The RF simulator uses the RF via parameters for simulation. RF via parameters are supported in the RF Shape Parameters Dialog Box. You must back annotate any RF via changes.

Figure 4-58 is a schematic view of two striplines connected by an RF via (VIA2). The stripline TL1 is on the top layer and stripline TL2 is on the bottom layer.

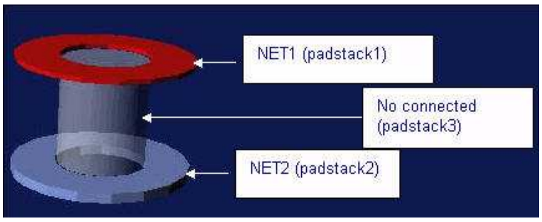
Figure 4-58. Schematic Capture RF Via



NET1 is the net between TL1 and the entry pad on VIA2. NET2 is the net between VIA2 exit pad and TL2. In layout NET1 and NET2 appear as one net. In the RF simulator, the nets are deconstructed into manageable pieces (NET1, NET2, and a drill hole with parameters).

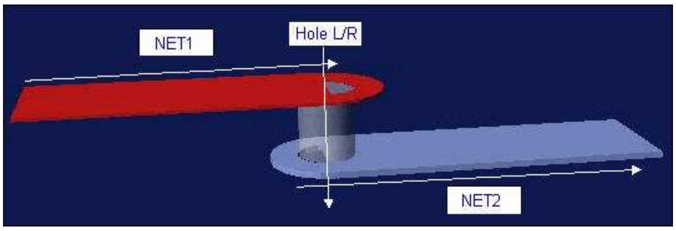
Figure 4-59 and Figure 4-60 show an RF simulator view of an RF via.

Figure 4-59. Layout RF Via



When you use the RF via to connect two striplines, the striplines and the RF vias entry/exit pads belong to different nets as in Figure 4-60.

Figure 4-60. Layout RF Via with Nets



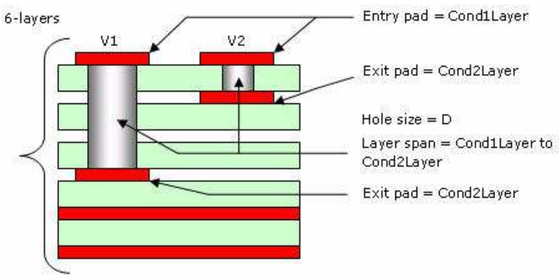
Common for all RF vias is the layer span. The layer on which the pad is visible is controlled by the RF via parameters. Drill holes are blind, buried, or through hole depending on the layer span.

The RF via parameters are available from the library_elements file as any other RF shape (*<project_dir>/RF/library_elements*).

The RF vias entry, exit, drill diameter, angle and material characteristics are common parameters for all RF vias (RF Control Dialog Box). The *Cond1Layer* parameter specifies the RF via entry pad. The *Cond2Layer* parameter specifies the RF via exit pad. The via hole size is controlled by the *D* parameter.

The drill hole only spans the entry and exit layer as defined by the *Cond1Layer* and *Cond2Layer* parameters. Figure 4-61 shows VIAFC RF vias with different parameters. For more information on RF vias, refer to the RF simulator documentation (TLines-Microstrip).

Figure 4-61. RF Via Layer Span



Related Topics

Vias in RF Designs

Standard Vias and RF Nodes

Creating and Modifying a Meander

Objects in RF Groups - Special Conditions

Changing an RF Net Name

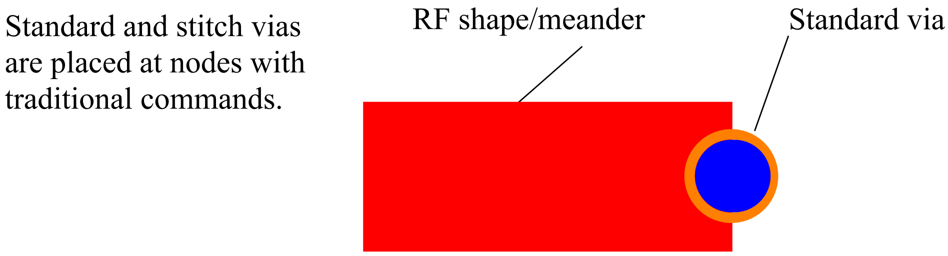
Place Via Dialog Box

Standard Vias and RF Nodes

A non-RF via (a via not in the schematic) is added in layout as any other via. Standard vias are able to connect directly to a node (Figure 4-62). The via pad overlaps the RF shape when the node and via have the same net.

You can only add the standard via when routing from the node or with the Place Via Dialog Box (with the RF Toolkit active, select **Edit** > **Place** > **Via**).

Figure 4-62. Regular Vias on Nodes



Related Topics

Vias in RF Designs

Creating and Modifying a Meander

Standard Vias and RF Nodes

Trace, Via, and Pad Clearances and Connections

Objects in RF Groups - Special Conditions

Changing an RF Net Name

Place Via Dialog Box

To access:

- With the RF Toolkit ( ) active, click **Place Vias** ( ) (RF Toolkit Toolbar)
- **Edit** > **Place** > **Via** (with RF Toolkit active)

Description

Place stitching vias around the contour of an object interactively or automatically with the Place Via dialog box.

Stitching vias create a ground shield around a radiating object. The vias are standard vias (Vias in RF Designs). The vias follow the contours of planes, conductive shapes, RF shapes and traces as defined by the settings in Place Via dialog box tabs (Table 4-18).

Table 4-18. Place Via Dialog Box Tabs Contents

Dialog Tab	Description
Place Via Dialog Box - Interactive Tab	The Interactive tab enables you to place regular vias interactively and define the span, net name, location, and placement conditions for the vias.
Place Via Dialog Box - Stitch Contour Tab	The Stitch Contour tab enables you to place rows of regular vias that follow the contour of a plane, conductive shape, trace, or RF shape.
Place Via Dialog Box - Stitch Shape Tab	The Stitch Shape tab enables you to fill a plane or conductive shape with regular vias.
Place Via Dialog Box - Radial Tab	The Radial tab enables you to place regular vias in a circular pattern with a start and end angle.
Place Via Dialog Box - Array Tab	The Array tab enables you to place an array of regular vias within a plane or conductive shape.

Place Via Dialog Box - Interactive Tab

Description

The Interactive tab enables you to place regular vias interactively and define the span, net name, location, and placement conditions for the vias.

Figure 4-63. Place Via Dialog Box - Interactive Tab

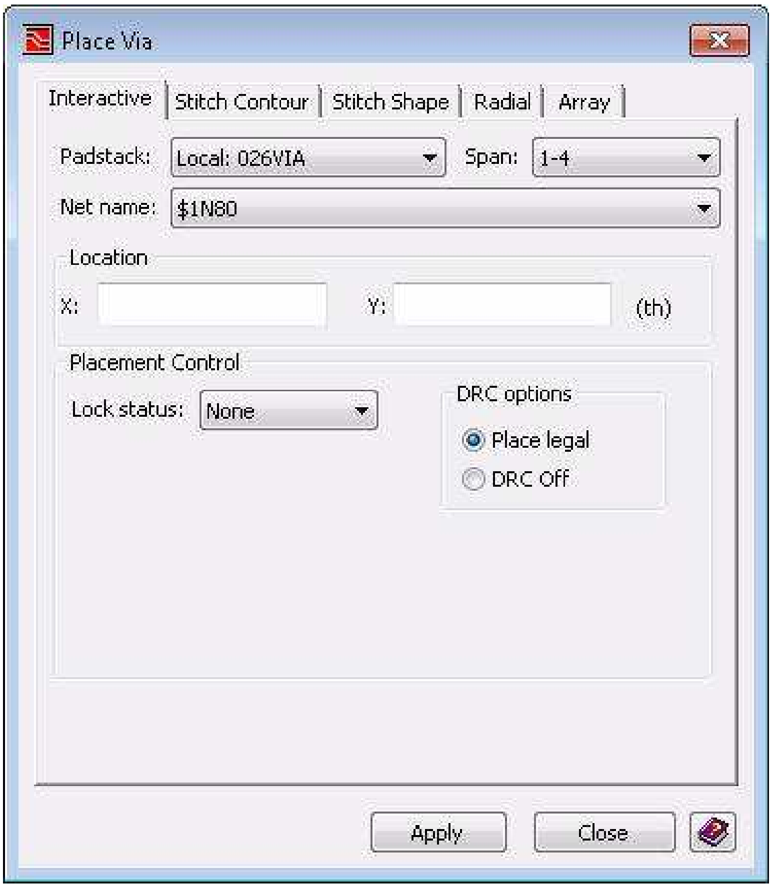


Table 4-19. Place Via Dialog Box - Interactive Tab Contents

Field	Description
Padstack	Defines the padstack to assign to the vias. For more information on creating padstacks, refer to Using Padstack Editor in Library Manager in the *Library Manager Process Guide*.
Span	Defines the layer span for the vias.
Net name	Defines the net name to assign to the vias based on the name of the plane or conductive shape. You can also populate the Net Name field by clicking a trace, pad, plane shape, RF node, or conductive shape that has an attached net.
Location	Defines the via location. Type the X and Y coordinates in the text boxes.

Table 4-19. Place Via Dialog Box - Interactive Tab Contents (cont.)

Field	Description
Lock status	Defines the via status. • **None** — Removes a lock or fix. • **Fixed** — Temporarily fixes the via location. • **Locked** — Permanently locks the via location.
DRC Options section	
Place legal	Selected, places only those vias that do not cause a DRC violation (DRC RF Violations and Design Rules Check).
DRC Off	Selected, turns off DRC checking, enabling the placement of all vias.

Place Via Dialog Box - Stitch Contour Tab

Description

The Stitch Contour tab enables you to place rows of regular vias that follow the contour of a plane, conductive shape, trace, or RF shape.

Figure 4-64. Place Via Dialog Box - Stitch Contour Tab

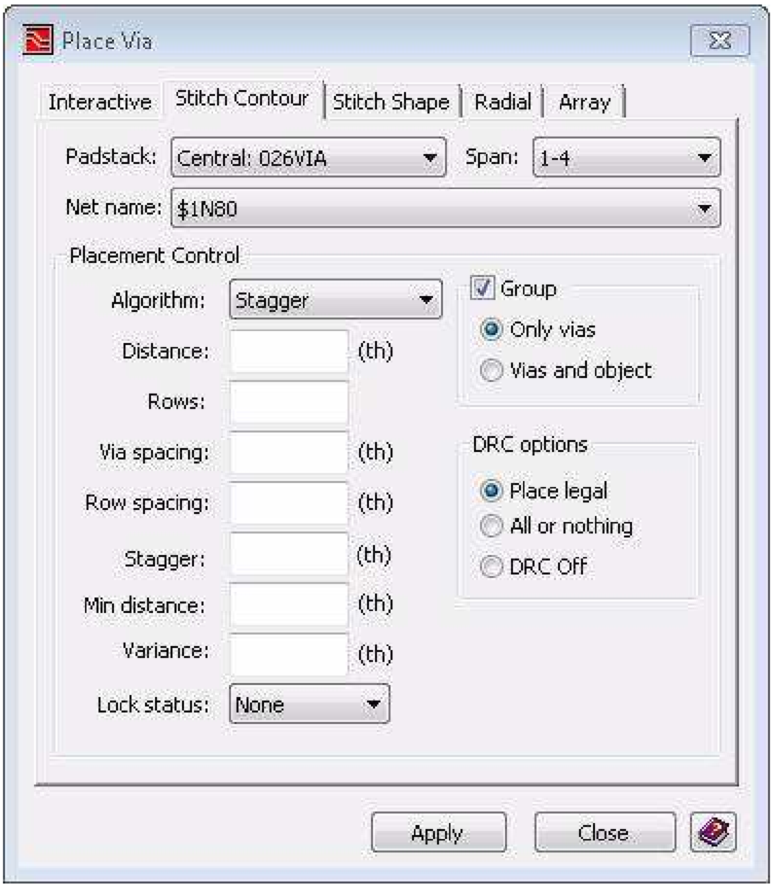


Table 4-20. Place Via Dialog Box - Stitch Contour Tab Contents

Field	Description
Padstack	Defines the padstack to assign to the vias. For more information on creating padstacks, refer to Using Padstack Editor in Library Manager in the *Library Manager Process Guide*.
Span	Defines the layer span for the vias.
Net name	Defines the net name to assign to the vias based on the net name of the plane or conductive shape. You can also populate the Net Name field by clicking a trace, pad, plane shape, RF node, or conductive shape that has an attached net.

Table 4-20. Place Via Dialog Box - Stitch Contour Tab Contents (cont.)

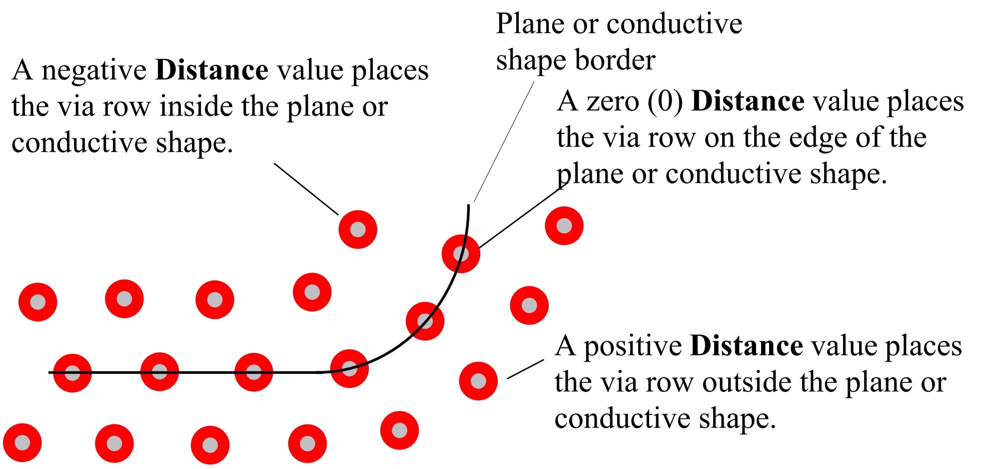
Field	Description
Placement Control section — Defines via placement parameters.	
Algorithm	Defines the placement algorithm. The placement of the first row of stitch vias is independent of the algorithm. The first row of stitched vias is placed using calculations based on the Distance and Via spacing values. The start and end points of the first stitch via row uses the minimum Via spacing value. Select a placement algorithm from the dropdown list: • Triangulation - 1 Algorithm — Creates two or more via rows with staggering vias that start with the second via row. • Triangulation - 2 Algorithm — Creates three or more via rows with staggering vias that start with the second via row and additional vias. • Stagger Algorithm — Creates two or more staggered via rows that are placed perpendicularly.
Distance	Defines the distance between the plane or conductive shape and the first row of the stitch vias. The **Distance** value can be: • **0** — Placement occurs on the edge of the plane or conductive shape. • **Positive** — Placement of the first via row occurs the **Distance** value outside the plane or conductive shape. • **Negative** — Placement of the first via row occurs the **Distance** value inside the plane or conductive shape. 
Rows	Defines the number of via rows.
Via spacing	Defines the center-to-center spacing between the vias of a row.
Row spacing	Defines the distance between the via rows.

Table 4-20. Place Via Dialog Box - Stitch Contour Tab Contents (cont.)

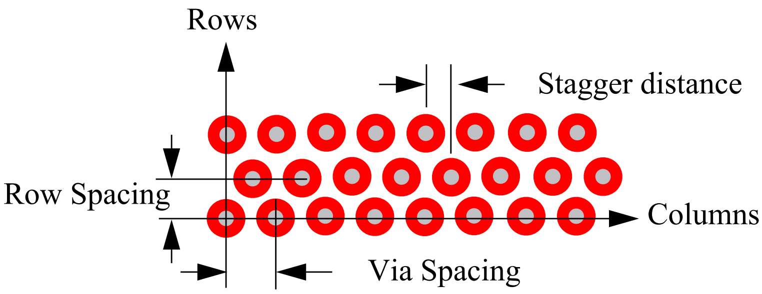
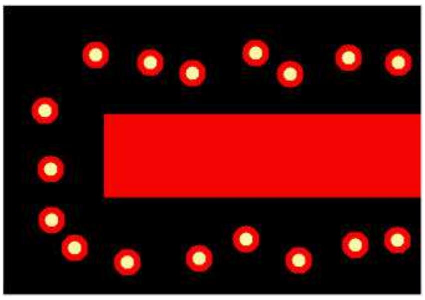
Field	Description
Stagger	Defines the stagger distance for the next via row. 
Min distance	Defines the minimum distance between vias in different rows. With more than one via row, the via spacing value only applies to the first row. For every consecutive via row, the via position is triangulated. If two vias are closer than the minimum distance, only one via is inserted.
Variance	Defines the value used to vary the via placements to avoid symmetrical via patterns. The variance value is randomly added to or subtracted from the via placement coordinates. All via rows are adjusted by the variance value. A variance value of zero does not alter the placement coordinates, and thus maintains a symmetrical via pattern. 
Lock status	Defines the via status. • **None** — Removes a lock or fix. • **Fixed** — Temporarily fixes the via location. • **Locked** — Permanently locks the via location.
Group section	
Only vias	Selected, creates a new RF group that contains only stitch vias. The center of the first via is the default reference point for the new RF group. If a new RF subgroup is created, it adopts the reference point of the main RF group to which it belongs.
Vias and objects	Selected, creates a new RF group that contains the stitch vias and the associated object. If the associated object is a member of an existing RF group, the vias become an RF subgroup of that RF group.

Table 4-20. Place Via Dialog Box - Stitch Contour Tab Contents (cont.)

Field	Description
DRC options section — Enables you to set Design Rule Check (DRC) options (Design Rules Check).	
Place legal	Selected, places only vias that do not cause a DRC violation (DRC RF Violations).
All or nothing	Selected, places all vias if no DRC violation occurs, otherwise does not place vias.
DRC Off	Selected, turns off DRC, enabling the placement of all vias.

Triangulation - 1 Algorithm

Use the **Triangulation - 1** algorithm to create two or more regular via rows. The second and subsequent via rows are staggered by half the **Via spacing** value. The via placement in the second and subsequent rows is perpendicular to the row spacing while maintaining the previous value (Figure 4-65 and Figure 4-66).

Figure 4-65. Triangulation - 1 Algorithm Display

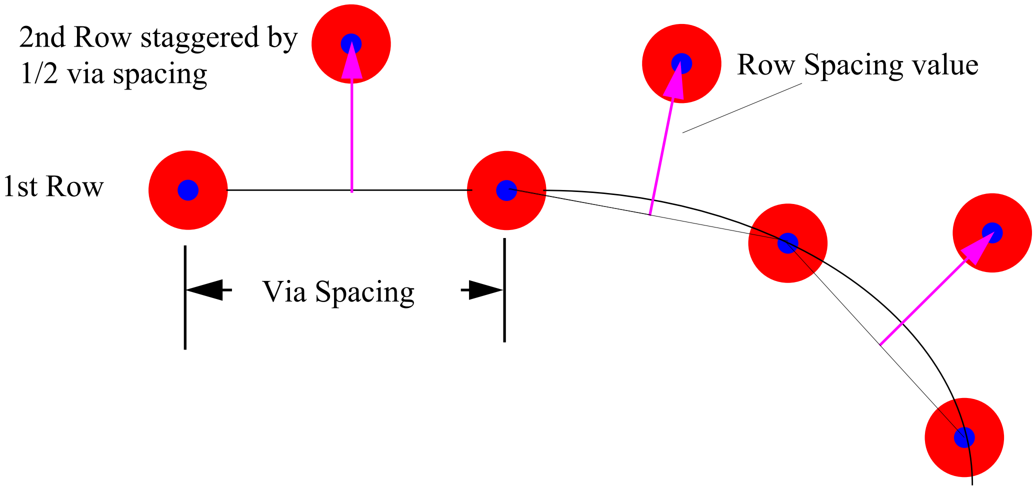
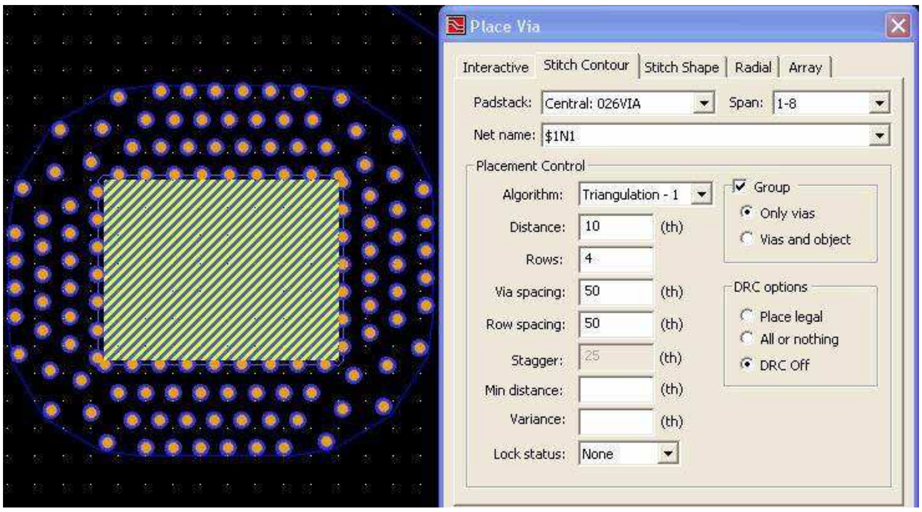


Figure 4-66. Triangulation - 1 Algorithm Row Display



Triangulation - 2 Algorithm

Use the **Triangulation - 2** algorithm to create three or more regular via rows. The second and following via rows are staggered by half the **Via spacing** value. The algorithm optimizes the number of vias by adding more vias if the distance between two existing vias is at least twice the **Via spacing** value (Figure 4-67 and Figure 4-68).

Figure 4-67. Triangulation - 2 Algorithm Display

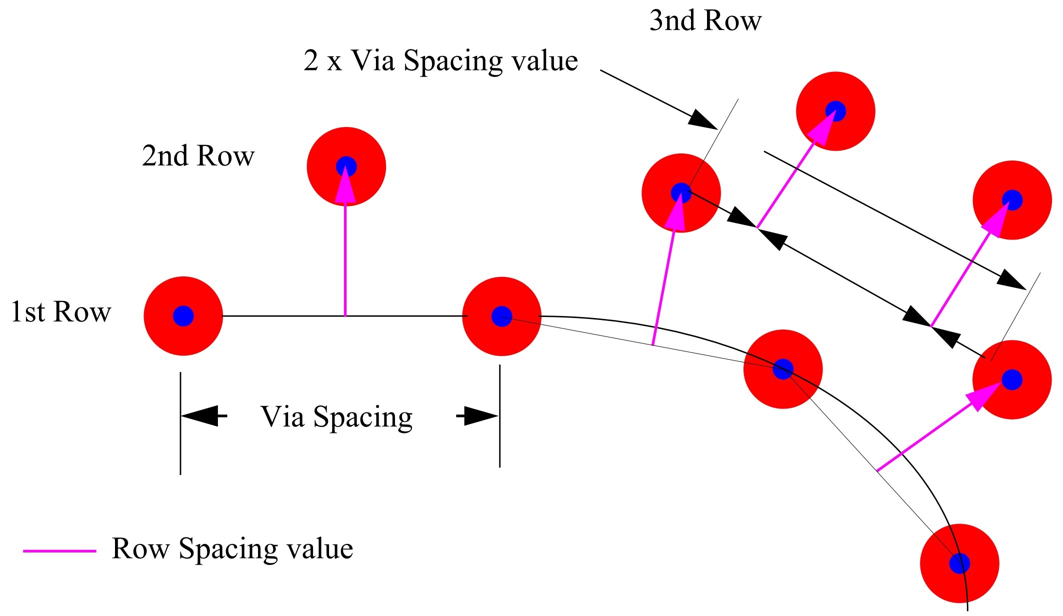
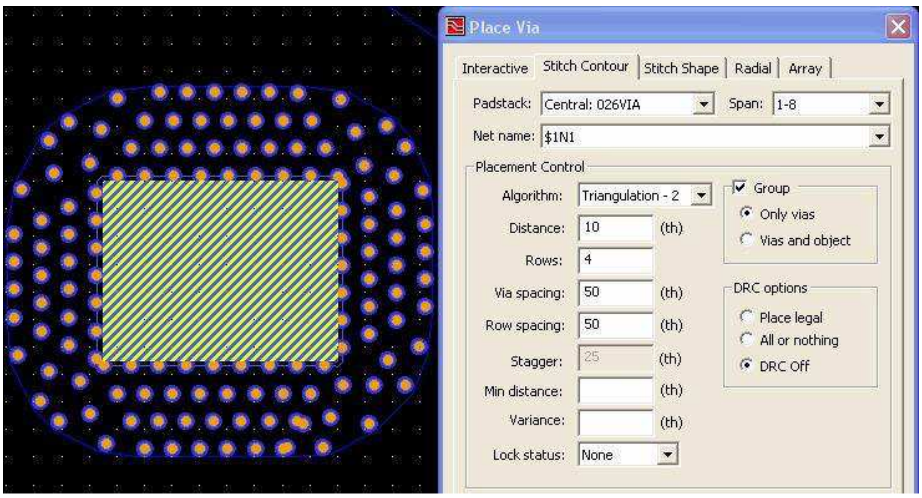


Figure 4-68. Triangulation - 2 Algorithm Row Display



Stagger Algorithm

Use the **Stagger** algorithm to create two or more regular via rows. The via placement is perpendicular to the row spacing while maintaining the **Row spacing** value. The vias in the second and subsequent rows are offset by the **Stagger** value in the direction parallel to an imaginary line connecting the two vias from the previous row.

For via placement around the corners of the second and subsequent rows, the algorithm places additional vias on an imaginary concentric arc derived from the arc formed by the two vias from the previous row. To find the via insertion point, the algorithm uses an intersection circle that is tangent to the previous arc with a radius equal to the **Via spacing** value (Figure 4-69 and Figure 4-70). The algorithm maintains the specified via spacing for all via placements.

Figure 4-69. Stagger Algorithm Display

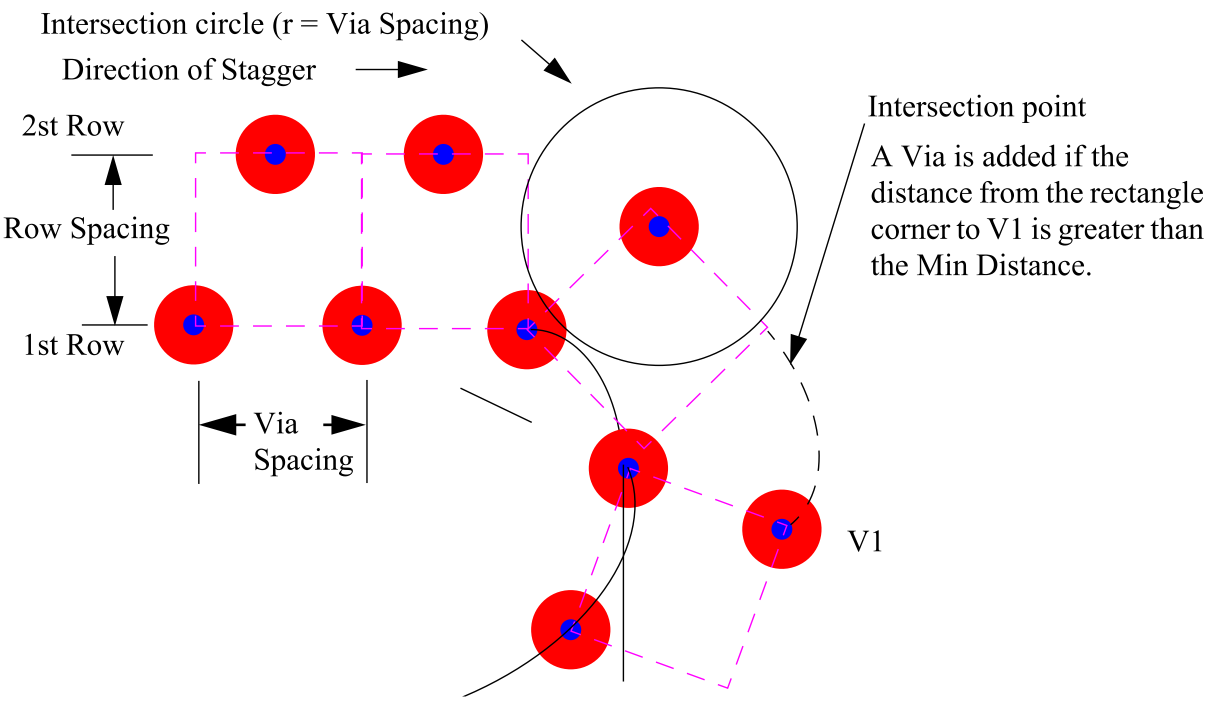
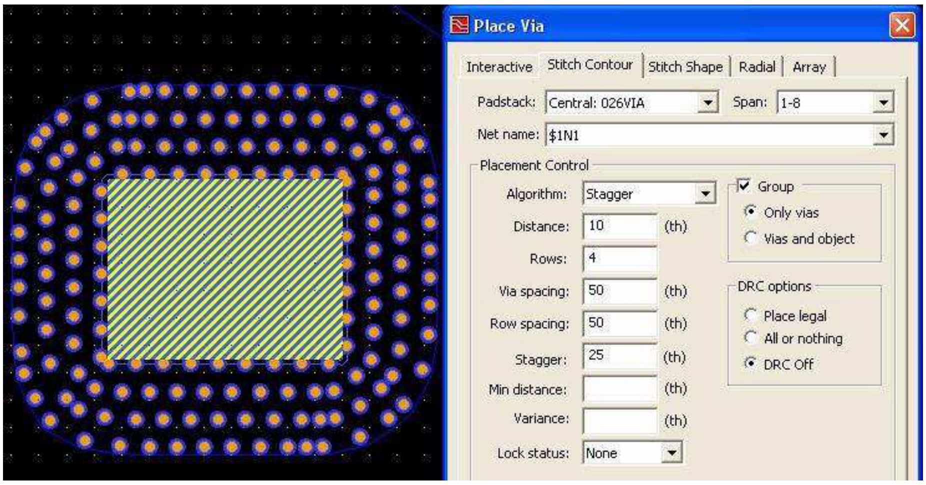


Figure 4-70. Stagger Algorithm Row Display



Place Via Dialog Box - Stitch Shape Tab

Description

The Stitch Shape tab enables you to fill a plane or conductive shape with regular vias.

Figure 4-71. Place Via Dialog Box - Stitch Shape Tab

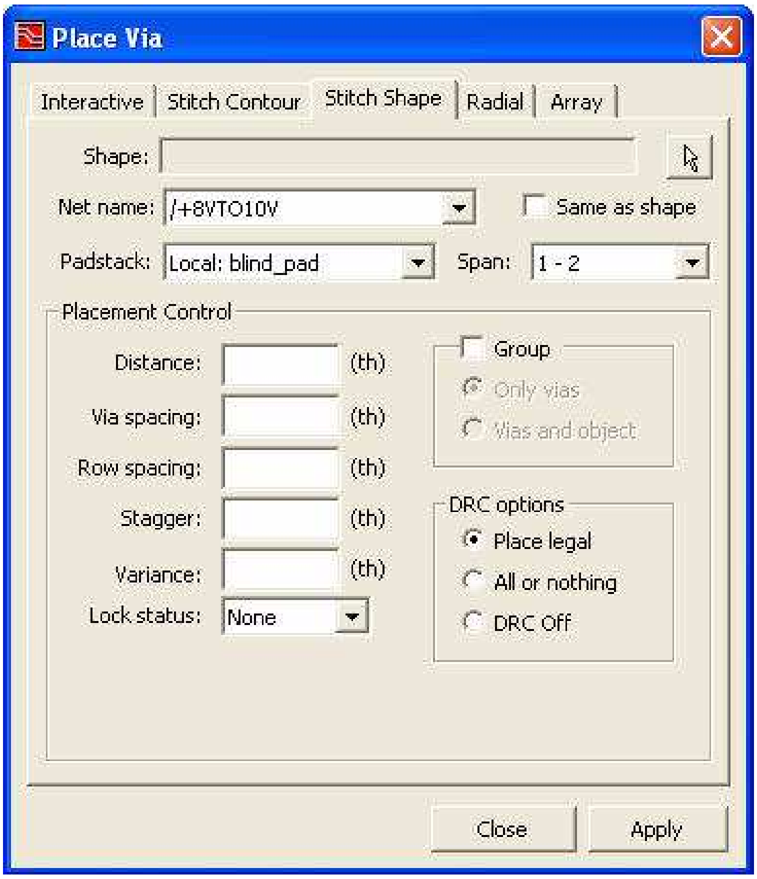


Table 4-21. Place Via Dialog Box - Stitch Shape Contents

Field	Description
Shape	Displays the name, location, and net of the plane or conductive shape selected by **Pick** (Example: User 1 at 113, -112 Net: ($1N2)). You cannot edit the **Shape** field.
Pick reference	Click, enables you to select a plane or conductive shape. Click **Pick reference** ( ), then click a plane or conductive shape to populate the **Shape** field.
Net name	Defines the net name to assign to the vias based on the name of the plane or conductive shape.

Table 4-21. Place Via Dialog Box - Stitch Shape Contents (cont.)

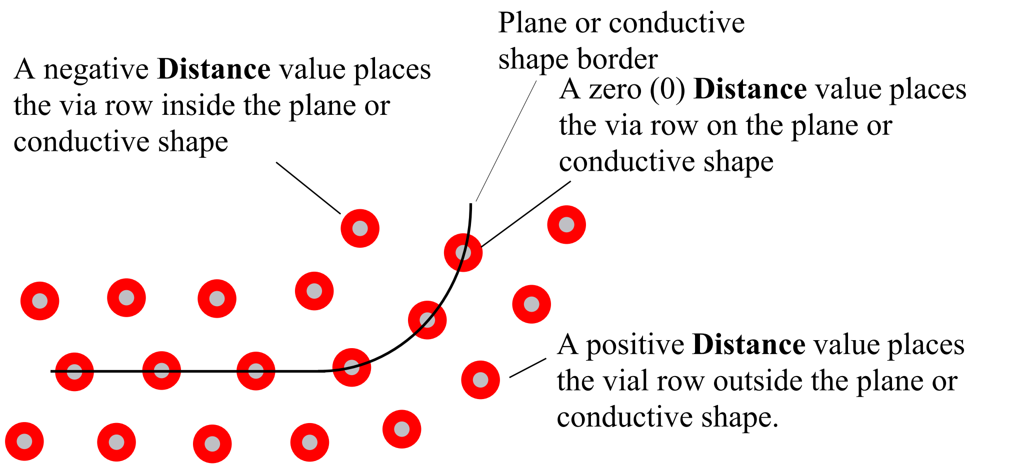
Field	Description
Same as shape	Checked, assigns the net name to the vias. Unchecked, assigns vias net name to the selection in the **Net name** field.
Padstack	Defines the padstack to assign to the vias. For more information on creating padstacks, refer to Using Padstack Editor in Library Manager in the *Library Manager Process Guide*.
Span	Defines the layer span for the vias.
Placement Control section — Defines the via placement parameters.	
Distance	Defines the distance between the plane or conductive shape and first row of the stitch vias. The **Distance** value can be: • **0** — Placement occurs on the edge of the plane or conductive shape. • **Positive** — Placement of the first via row occurs the Distance value away from the plane or conductive shape. • **Negative** — Placement of the first via row occurs the Distance value inside the plane or conductive shape. 
Via spacing	Defines the center-to-center spacing between the vias of a row.
Row spacing	Defines the distance between the via rows.

Table 4-21. Place Via Dialog Box - Stitch Shape Contents (cont.)

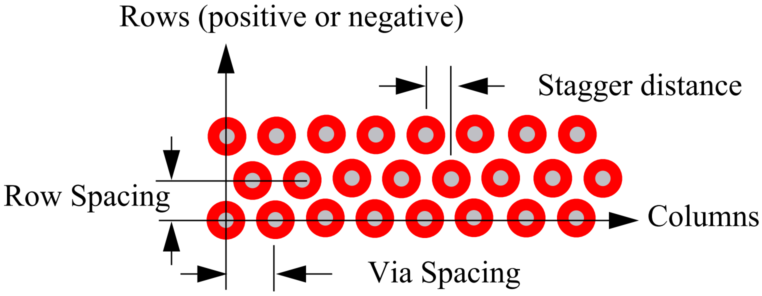
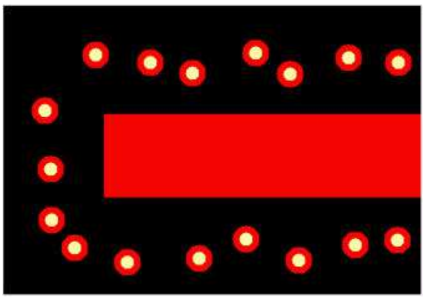
Field	Description
Stagger	Defines the stagger distance for the next via row. 
Variance	Defines the value used to vary the via placements to avoid symmetrical via patterns. The variance value is randomly added to or subtracted from the via placement coordinates. All via rows are adjusted by the variance value. A variance value of zero does not alter the placement coordinates, and thus maintains a symmetrical via pattern. 
Lock status	Defines the via status. • **None** — Removes a lock or fix. • **Fixed** — Temporarily fixes the via location. • **Locked** — Permanently locks the via location.
Group section	
Only vias	Selected, creates a new RF group that contains only the stitched vias. The default RF group reference point is at the center of the first via inserted. **Note**: The new via RF group uses the center of the RF group as the default reference point. The new RF subgroup uses the RF group reference point.
Vias and object	Selected, creates a new RF group that contains the stitched vias and the associated object. If the object is an RF group member, the vias become an RF subgroup of that RF group. **Note**: The new via RF group uses the center of the RF group as the default reference point. The new RF subgroup uses the RF group reference point.

Table 4-21. Place Via Dialog Box - Stitch Shape Contents (cont.)

Field	Description
DRC options section — Enables you to choose a Design Rule Check (DRC) option (Design Rules Check).	
Place legal	Selected, places only vias that do not cause a DRC violation (DRC RF Violations).
All or nothing	Selected, places all vias if no DRC violation occurs, otherwise does not place vias.
DRC Off	Selected, turns off DRC, enabling the placement of all vias.

Place Via Dialog Box - Radial Tab

Description

The Radial tab enables you to place regular vias in a circular pattern with a start and end angle.

Figure 4-72. Place Via Dialog Box - Radial Tab

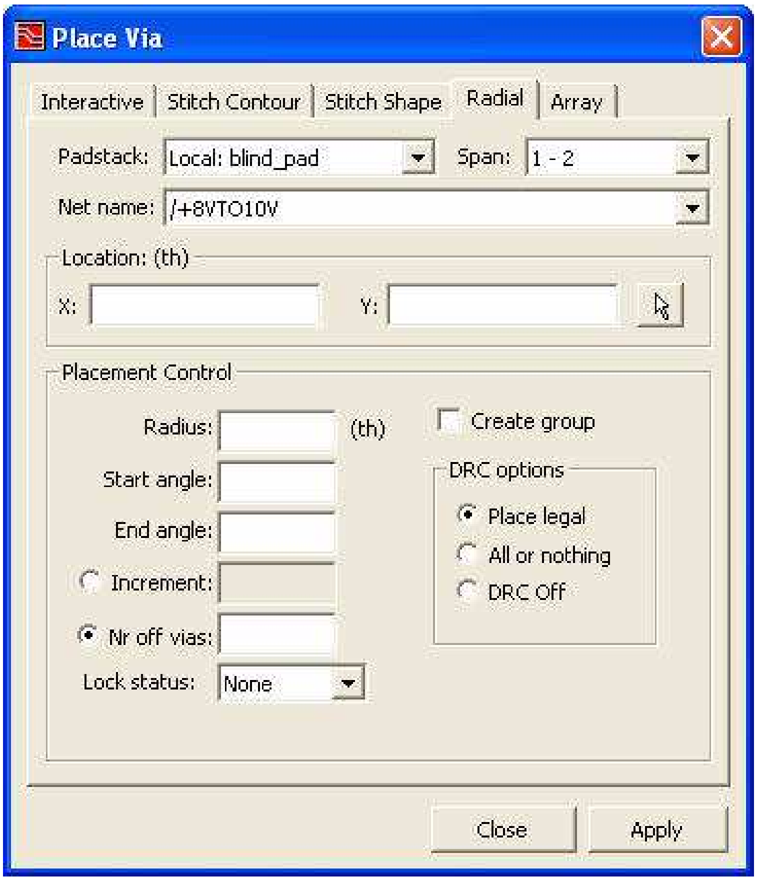


Table 4-22. Place Via Dialog Box - Radial Contents

Field	Description
Padstack	Defines the padstack to assign to the vias. For more information on creating padstacks, refer to Using Padstack Editor in Library Manager in the *Library Manager Process Guide*.
Span	Defines the layer span for the vias.
Net name	Defines the net name to assign to the vias based on the net name of the plane or conductive shape. Select the net name from the dropdown list. You can also populate the Net Name field by clicking on a trace, pad, plane shape, RF node or conductive shape that has an attached net.

Table 4-22. Place Via Dialog Box - Radial Contents (cont.)

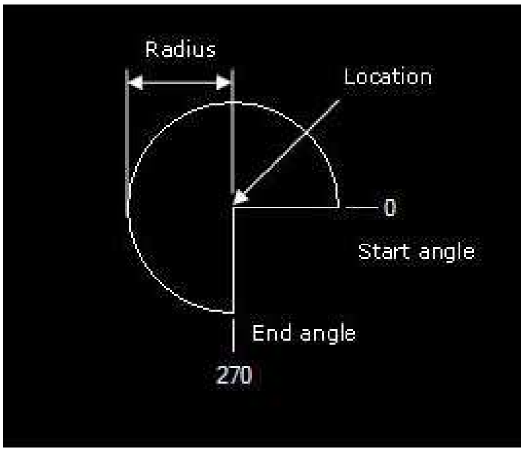
Field	Description
Location section — Enables you to define the center of the circle.	
X, Y	Defines the center location of the placement circle. Enter a location or use the **Pick** option.
Pick reference	Click, enables you to select a location or object in the layout tool, which populates the Location fields. Click **Pick reference** ( ) and then select a location or object in layout.
Placement Control section — Defines via placement parameters.	
Radius	Defines the distance of the placement radius. 
Start angle	Defines the start angle, which determines the location of the first via on the arc or circumference. You can leave the Start and End angle blank for prompting. Create a full circle with a start angle 0° and an end angle 360°.
End angle	Defines the end angle, which determines the location of the last via on the arc or circumference. You can leave the Start and End angle blank for prompting. Create a full circle with a start angle 0° and an end angle 360°.
Increment	Selected, defines the angle increment for placing vias on the arc or circumference. If you select **Increment** you cannot select **Nr of vias**.
Nr of vias	Selected, defines the number of vias to place on the arc or circumference. Distributes the number of vias evenly. If you select **Nr of vias** you cannot select **Increment**.
Lock status	Defines the via status. • **None** — Removes a lock or fix. • **Fixed** — Temporarily fixes the via location. • **Locked** — Permanently locks the via location.
Create group	Checked, creates an RF group for the inserted vias. The RF group default name is: radialVias<*n*>, where <*n*> is the next available integer. The default RF group reference point is the center of the first via inserted.

Table 4-22. Place Via Dialog Box - Radial Contents (cont.)

Field	Description
DRC options section — Enables you to choose a Design Rule Check (DRC) option (Design Rules Check).	
Place legal	Selected, places only those vias that do not cause a DRC violation (DRC RF Violations).
All or nothing	Selected, places all vias unless a DRC violation occurs, in which case the via placement process aborts.
DRC Off	Selected, turns off DRC checking, enabling the placement of all vias.

Place Via Dialog Box - Array Tab

Description

The Array tab enables you to place an array of regular vias within a plane or conductive shape.

Figure 4-73. Place Via Dialog Box - Array Tab

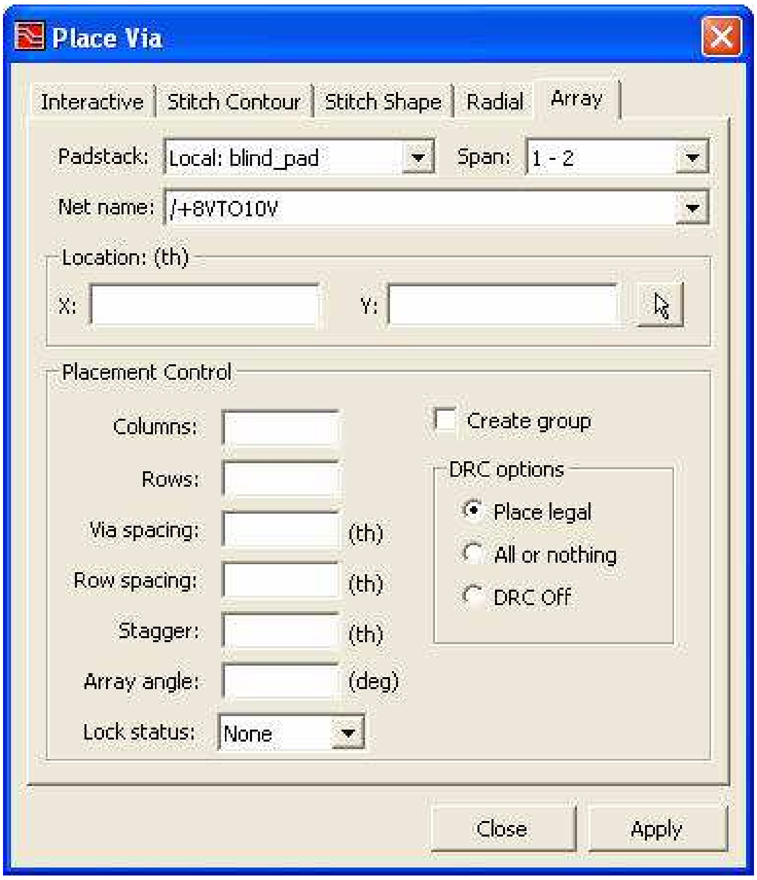


Table 4-23. Place Via Dialog Box - Array Contents

Field	Description
Padstack	Defines the via padstack. Select a padstack from the dropdown list. For more information on creating padstacks, refer to Using Padstack Editor in Library Manager in the *Library Manager Process Guide*.
Span	Defines the via layer span. Select a span from the dropdown list.
Net name	Displays the net name of the array. Select a net name from the dropdown list. You can also populate the **Net name** field by clicking on a trace, pad, plane shape, RF node or conductive shape that has an attached net.
Location section — Enables you to define the start location of the array.	
X, Y	Defines the start location of the array. Enter the X, Y coordinates for the location point or use the **Pick** option.

Table 4-23. Place Via Dialog Box - Array Contents (cont.)

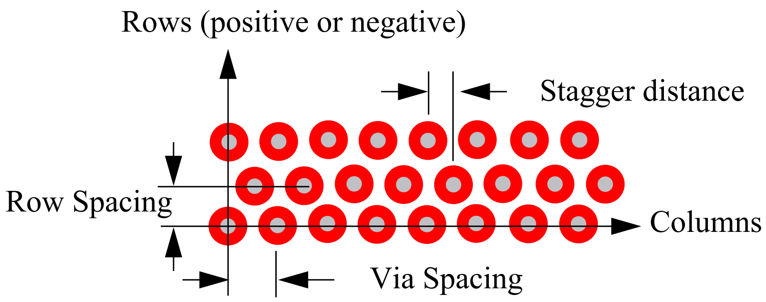
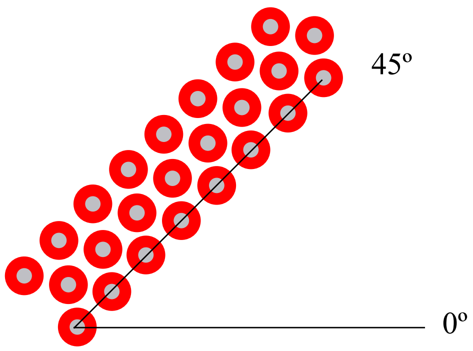
Field	Description
Pick reference 	Click, enables you to select a location or object in the layout, which populates the **Location** fields. Click **Pick reference** () and then select a location or object in layout.
Placement Control section — Defines the via placement parameters.	
Columns	Defines the number of via columns.
Rows	Defines the number of via rows.
Via spacing	Defines the center-to-center spacing between the vias of a row.
Row spacing	Defines the distance between the via rows.
Stagger	Defines the stagger distance for the next via row. 
Array angle	Defines the angle of rotation (counter-clockwise). 
Lock status	Defines the via status. • **None** — Removes a lock or fix. • **Fixed** — Temporarily fixes the via location. • **Locked** — Permanently locks the via location.
Create group	Checked, creates an RF group containing the inserted vias. The default RF group name is: arrayVias<*n*>, where <*n*> is the next available integer. The default RF group reference point is at the center of the first via inserted.

Table 4-23. Place Via Dialog Box - Array Contents (cont.)

Field	Description
DRC options section — Enables you to choose a Design Rule Check (DRC) option (Design Rules Check).	
Place legal	Selected, places only those vias that do not cause a DRC violation (DRC RF Violations).
All or nothing	Selected, places all vias if no DRC violation occurs, otherwise does not place vias.
DRC Off	Selected, turns off DRC, enabling the placement of all vias.

Related Topics

Creating and Modifying RF Shapes

Editing Existing RF Shapes

Auto Arranger

To access:

- **Setup > RF > Auto Arranger**
- RF Shape Parameters Dialog Box
- With RF Toolkit active, choose **Place > Place Parts and Cells** and then choose Automatic from the Method field dropdown list (Placing RF Groups with Auto Arranger).
- **RF Control > RF Design** tab, select an RF group and choose Auto Arrange (RF Control Dialog Box).
- With the RF Toolkit active, choose **RF > Auto Arranger**.

The auto arranger command automatically moves RF shapes and non-RF shapes in the selected RF group using connectivity, angle, node to node, and node to pad rules.

Moving RF shapes or parts and changing RF shape parameters may disconnect the RF shapes and parts. Use the Auto Arranger to reconnect the RF shapes and parts.

The following applies when you move or connect shapes/parts with Auto Arranger:

Invoke auto arranger with nothing selected — Select the shape or part seed when prompted. The seed point remains stationary and all members of the RF group move to meet the rules.

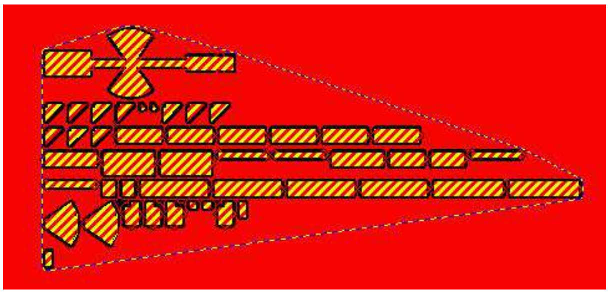
Invoke auto arranger with one shape/part selected — Selection of the shape or part becomes the seed point. All members of the RF group move to meet the rules, while the seed remains stationary.

Invoke auto arranger with 2 or more shapes/parts selected — Select the shape or part seed when prompted. The seed point remains stationary and only the selected shapes/parts connect to the seed if the rules apply.

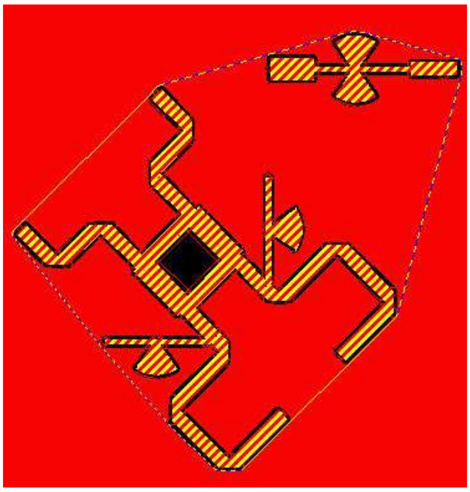
Figure 4-74 shows an RF group imported from schematic capture and the results of placing the RF group in layout with the automatic selection in the Place Parts and Cells dialog box (Placing RF Groups with Auto Arranger).

Figure 4-74. Auto Arrange Before and After

Forward Annotation of an RF group



Automatic selection results

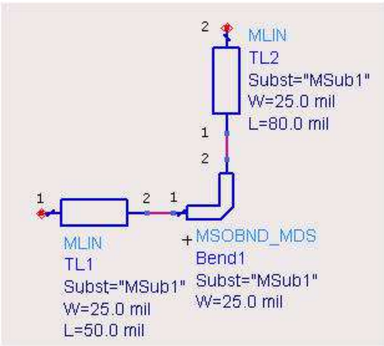


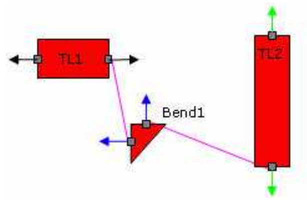
The RF group maintains the relative angle between the connected RF shapes or parts (members). The relative angle maintains the RF group member position during moves and rotation (Replacement and Repair).

Figure 4-75 shows the basic functionality of the Auto Arranger on a node-to-node circuit having two microstrip lines and one microstrip bend (RF Entry Rules Dialog Box).

Figure 4-75. RF Circuit Schematic to Layout

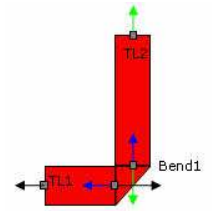
Schematic Capture **Layout**





Auto Arranger Results

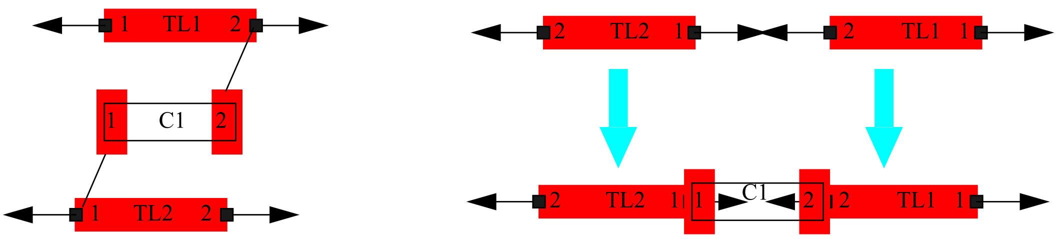
TL1 is the seed shape.



The Auto Arranger uses the default node-to-node angle rules for initial placement.

Nodes determine the placement and connectivity for regular parts in an RF group (RF Shape Node Types).

Figure 4-76. RF Circuit Layout - Default Nod-to-Pad Rules



C1 is the seed, TL1 and TL2 rotate 180° to maintain node-to-pad rules.

Related Topics

Placing RF Groups with Auto Arranger

Using RF Groups

Changing Snap Points

Creating and Modifying RF Shapes

RF Shape Parameters Dialog Box

To access:

- With the RF Toolkit ( ) active, click **Parametric Parameters** ( ) (RF Toolkit Toolbar)
- Double-click an RF shape
- **Setup > RF > RF Parameters**
- Select an RF shape and choose **RF > RF Parameters**

Description

Alter RF shapes and define Var block variable parameters with the RF Shape Parameters dialog box.

You can use the Var block properties to define other RF shape parameters (Adding a Varblock). Parameter changes are transferred to the schematic capture tool during back annotation.

Figure 4-77. RF Shape Parameters Dialog Box

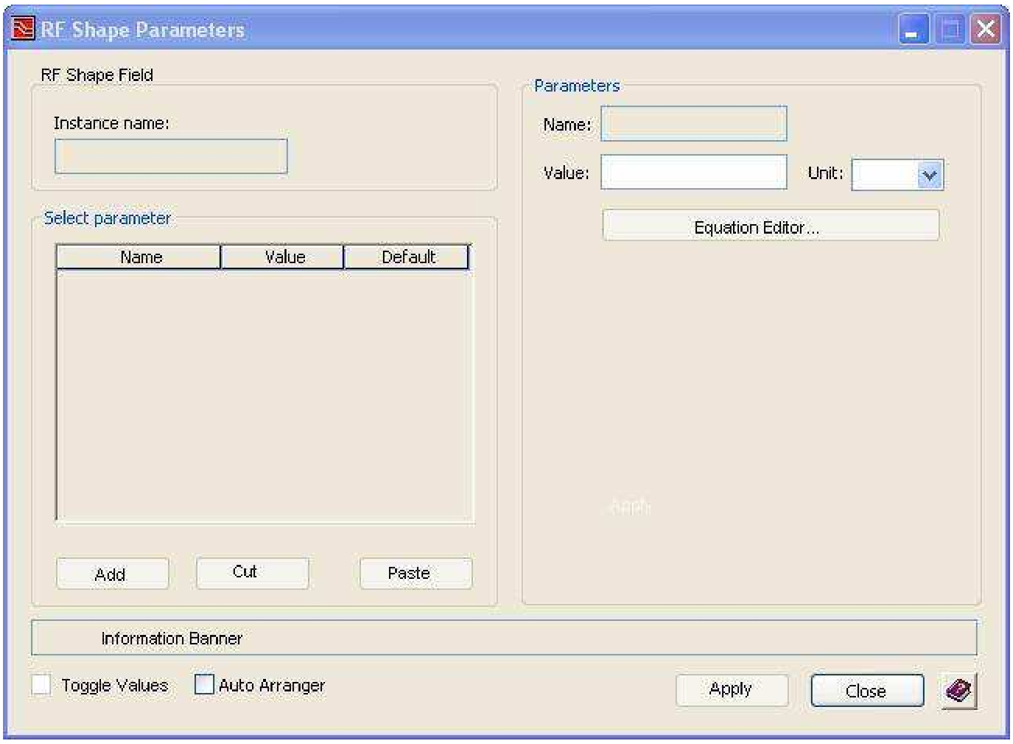


Table 4-24. RF Shape Parameters Dialog Box Contents

Field	Description
Instance name	Displays the reference designator of the RF shape or the meander name. If you select more than one RF shape, this field displays the number of parts selected (Example: 2 parts selected).
Select parameter	Displays the RF shape parameters for the selected object, which enables you to change the values. Change parameter values by selecting the parameter and changing the value in the **Parameters Value** field. • RF_REGION and Subst have dropdown lists. • Changing the RF_REGION, removes the RF shape from the RF group and moves the RF shape to the chosen RF_REGION (RF group). **Note:** You cannot delete or add properties to an RF shape. You can only change the active parameters. Refer to the RF simulator documentation for descriptions of RF shape parameters.
Add	Adds a variable name and value to the Var block (Creating Var Block Variables).
Cut	Removes the selected variable from the Var block (Creating Var Block Variables).
Paste	Restores the Cut Var block variable (Creating Var Block Variables).
Parameters section — Enables parameter changes to RF shapes and the creation of variables in the Var block (Var Block).	
Name	Displays the name of the selected parameter (refer to Meander Segment and the RF simulator documentation for parameter descriptions). Defines the variable name. The **Name** field accepts all alphanumeric characters (a-z, A-Z, 0-9) with no spaces.
Value	Defines the parameter value. You can change the value as required: • Real numbers must be in the format 0.0. • Expressions that require evaluation must be in parentheses “()” (Var Block Calculations). • Supported arguments are listed in Equation Editor Expressions.
Unit	Defines the parameter units.
Equation Editor	Opens the RF Equation Editor Dialog Box.
Information Banner	Displays the selected RF shape parameter description.

Table 4-24. RF Shape Parameters Dialog Box Contents (cont.)

Field	Description
Toggle Values	Checked, displays the parameter **Value** in decimal point format (Example: 30.000000mil). Unchecked, displays the parameter **Value** in a format you can edit (Example: 30mil).
Auto Arranger	Checked, automatically arranges the RF group using connectivity, angle rule, and changes to RF shape parameters (Auto Arranger).

Related Topics

Editing Existing RF Shapes

Changing RF Shape Parameter Values

Using RF Groups

Replacement and Repair

Creating and Modifying RF Shapes

Use the following to create and modify RF shapes:

- Changing RF Shape Parameter Values
- Creating a Substrate Automatically
- Creating a Custom RF Shape
- Connecting Nodes and Pads with an RF Shape
- Replacement and Repair
- Creating a Custom RF Shape

Related Topics

Using RF Groups

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using Variables

Editing Existing RF Shapes

Changing RF Shape Parameter Values

RF shapes may require parameter changes during layout development.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Parametric Parameters** ( ).
3. Select an RF shape.
4. Select the parameter you want to change.

   The Parameters field is populated with the Value selected.
5. Alter the Parameters Value field as needed.

   On the RF _REGION and Subst parameters, use the Value dropdown list to choose your selection.
6. Click **Apply**.

Related Topics

Replacement and Repair

Using RF Groups

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Creating and Modifying RF Shapes

Adding a Varblock

Using Variables

Editing Existing RF Shapes

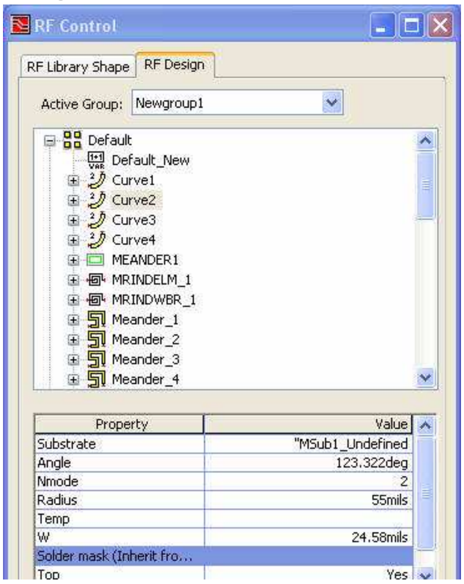
Creating a Substrate Automatically

The layout tool automatically creates a substrate when:

- A push occurs on a component that requires a substrate.
- A component resides on a load that does not match its substrate parameters.

When you create a substrate automatically, the components Substrate value becomes MSub1_undefined (Figure 4-78). The **Analysis > Review Hazards > Batch > RF Violations** provides additional information.

Figure 4-78. Substrate Value



Prerequisites

- Forward Annotation has occurred (**Setup > Project Integration**)
- A MSub1_undefined substrate is available.

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Parametric Properties** ( ).
3. Click the Substrate value (MSub1_undefined) dropdown arrow.
4. From the dropdown list, choose a Substrate value.

   If no substrate is applicable, create a substrate and use it for the Substrate value (Substrates).
5. Click **Apply**.

Results

The undefined substrate is defined.

Related Topics

Replacement and Repair	Adding a Varblock
Using RF Groups	Using Variables
Creating a Library Part	Editing Existing RF Shapes
Creating and Modifying RF Shapes	Creating a Custom RF Shape

Creating a Custom RF Shape

Create custom RF shapes with meanders and RF shapes.

When you convert an RF shape to a custom RF shape, a reference designator is automatically generated as *referencing layer 1*. The physical RF shape does not change to layer 1, only the reference layer for the RF shape is layer 1.

Note

If you apply boolean operators to the conductive shape in Draw mode, it loses its relation to the RF shape. Use the **Convert RF Shape** command to re-associate the conductive shape to an RF shape.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Create a conductive shape on layer 1. Use the **Draw mode** or import a conductive shape.
3. Create an RF group, not a subgroup with the new part name (Adding an RF Group or RF Subgroup).
4. Select the conductive shape.
5. Click **Convert RF shape** ().

   The RF shape becomes a member of the active RF group. The RF shape requires one or more nodes.
6. Add RF nodes to the conductive RF shape:
   - Adding an Edge Node
   - Adding a Floating Node
   - Adding a Stub

7. Set all RFPort pins to True.
   a. Click the **Control pane** ( ).
   b. Click the RF Design tab.
   c. Expand the RF group containing the RF shape.
   d. Expand the RF shape and select a pin.
   e. Use the dropdown menu to set the RFPort pin True.
8. Generate RF shape library symbol (Creating a Library Part).
9. Delete the custom RF shape in the layout tool.
10. Place the RF custom symbol in the schematic (Placing RF Symbols).
11. Forward annotate the schematic capture design (Layout tool - **Setup** > **Project Integration**).
12. Place the custom RF shape in the layout tool (Placing RF Groups with Auto Arranger).

Results

The custom RF shape becomes part of the layout design.

Related Topics

Replacement and Repair	Adding a Varblock
Using RF Groups	Using Variables
Creating a Library Part	Editing Existing RF Shapes
Creating and Modifying RF Shapes	Creating a Custom RF Shape

Connecting Nodes and Pads with an RF Shape

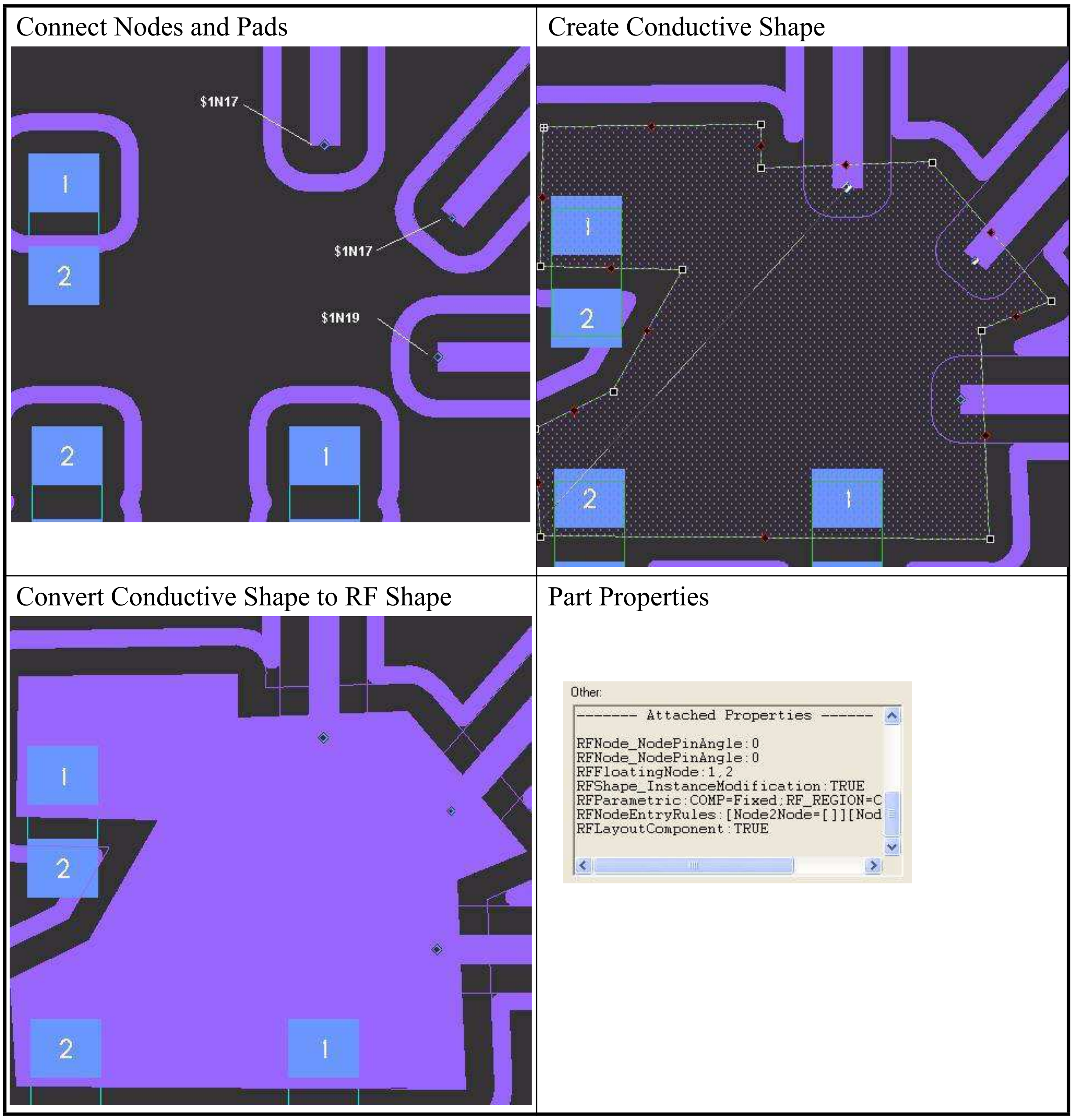
All the pads and nodes of a conductive RF shape create an electrical RF net (Multiple Nets and Pad Connections). Nodes and pads with different nets are connected with the RF shape (Multiple Nets and Pad Connections).

Procedure

1. Create a conductive shape that encloses the pads and nodes as shown in Figure 4-79.
2. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
3. Select the conductive shape.
4. Click **Convert RF shape** ( ).

   The RF shape becomes a member of the active RF group.

Figure 4-79. Conductive Shape and Nodes



5. Review that the Part Properties dialog box, Other field lists the RF shape nodes.

Related Topics

Replacement and Repair	Adding a Varblock
Using RF Groups	Using Variables
Creating a Library Part	Editing Existing RF Shapes

Creating and Modifying RF Shapes   Creating a Custom RF Shape

Replacement and Repair

After placing an RF group, you change the size of an RF shape by interactive editing or by updating parameters. If any changes break a circuit connection, use one of the following methods to repair the circuit.

- Choose **RF > Auto Arranger** and select the seed.
- Select **Setup > RF > Auto Arranger**.
- Select the RF in the RF Control Dialog Box, (**Control pane** ( )), select the RF group and choose Auto Arranger.

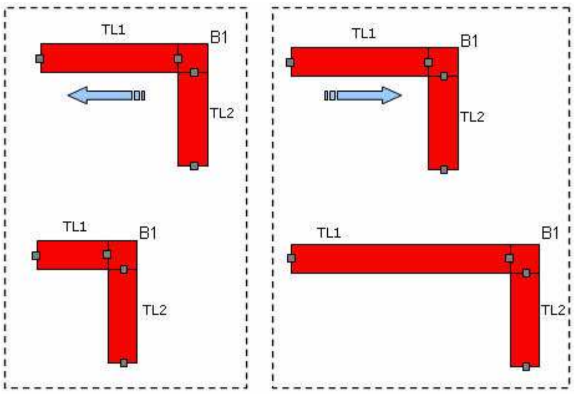
The Auto Arranger stretches, reduces, and shoves the RF shapes to meet the parametric changes with the RF shapes connectivity and angle rules.

When you invoke Auto Arranger and multiple RF shapes are involved, you are prompted to select the seed part. If the circuit **cannot** be repaired due to violations, the Auto Arranger terminates.

To prevent Auto Arranger termination, set the interactive Design Rule Check (DRC) to off. Use the **Analysis > Batch DRC** to run DRC (DRC RF Violations).

In Figure 4-80 TL1 is the seed RF shape for both sides because it is the shape being edited. In the left side, TL1 length is decreased by half the original length. When you invoke the Auto Arranger, it moves B1 and TL2 to maintain the connectivity and angle rules. In the right side, TL1 length is increased by half its length. When you invoke the Auto Arranger, it moves B1 and TL2 to maintain the connectivity and angle rules.

Figure 4-80. Repair Circuit after Shape Edit



Related Topics

Auto Arranger	Editing Existing RF Shapes
Using RF Groups	Creating a Custom RF Shape
Creating a Library Part	Creating and Modifying RF Shapes

Creating a Library Part

During project development you may need to create an RF shape and place it in the schematic capture design. Create a custom RF shape in the layout tool or import an RF shape from the RF simulator into the layout tool and place it in the library. You can instantiate the custom RF shape in the schematic capture tool as you do with any other RF shape.

Note

If you are using Agilent ADS RF simulator to create the RF shape, you need to add the ADS_MODEL property to the part. You add the ADS_MODEL_LIBRARY property for vendor RF parts that are outside the current project directory.

Custom RF shapes can contain the following:

- Conductive shapes
- Plane obstructs
- Stitch contour property
- User vias (with component vias)
- Fixed shapes
- Plane shapes
- Trace obstructs
- Via obstructs
- Pad obstructs
- Standard RF shapes
- Trace-via obstructs

Note

The custom RF shape is project specific. To use this shape in a different design, copy the library_elements file to the new project (<*project_dir*>*/RF/library_elements*).

Prerequisites

- The RF shape must be part of an RF group (Group/Ungroup Dialog Box).
- The RF shape must have at least one node (RF Shape Node Types).
- The RFPort property must be set to **True** for each RF shape node (RF Control Dialog Box, RF Design tab).

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).

2. Click **Control pane ( )**.
3. Select the **RF Design** tab.
4. Select the RF group that contains the new library part.
5. Choose **Generate Library Symbol**.

   The Create Schematic Symbol dialog box displays (Figure 4-82).

   If the RF group contains unsupported elements, a warning message displays.
6. Fill in the information in the Create Schematic Symbol Dialog Box.
7. Click **OK**.

   The custom RF shape definition is stored in the RF library elements file and the schematic symbol is stored in *...\SymbolLibs\<partition>\sym* in the library.

Results

The RF shape is listed in the RF Control Dialog Box, RF Library Shape tab - User folder.

Related Topics

Using RF Groups

Creating and Modifying RF Shapes

RF Shape Clearance Checking

Editing Existing RF Shapes

Creating a Custom RF Shape

Equation Editor Dialog Box

To access: With the RF Toolkit () active, click **Parametric properties** > Select parameter > Click **Equation Editor** (RF Shape Parameters Dialog Box)

Description

Use this dialog box to create variables and to create an RF shape parameter equation.

Refer to Equation Editor Expressions for supported functions.

Refer to the RF simulator documentation for more information on mathematical operators.

Note
Only real numbers are supported.

Figure 4-81. Equation Editor Dialog Box

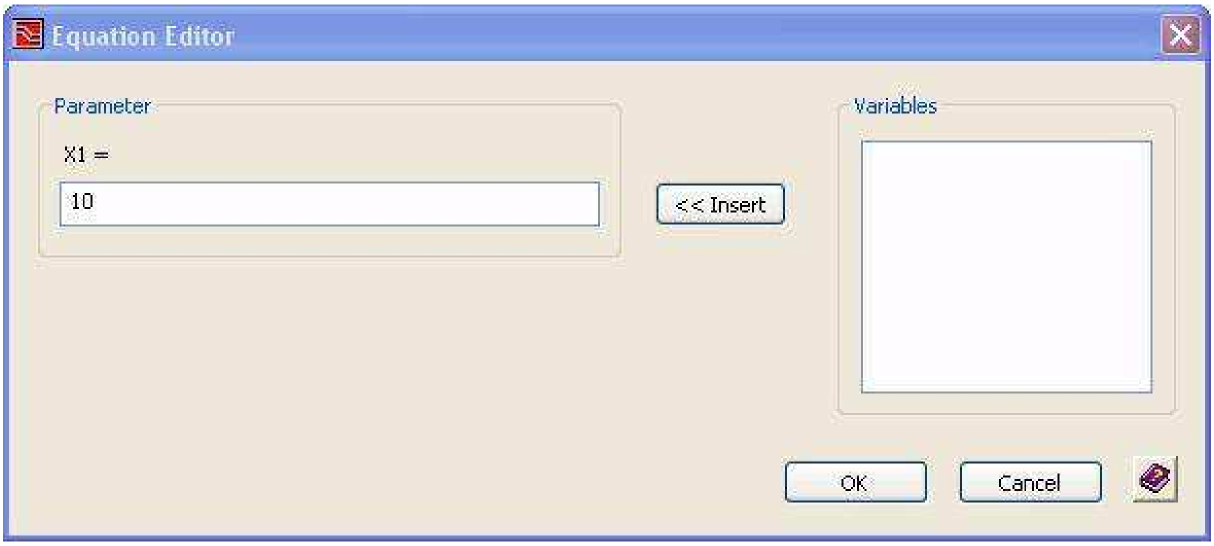


Table 4-25. Equation Editor Dialog Box Contents

Field	Description
Parameter	Defines the variables or parameter equation (Var Block Calculations).
<< Insert	Copies the selected variable in the **Variables** field (Var block variables) to the **Parameter** field.
Variables	Displays the Var block variables.

Related Topics

Using Variables

Var Block

Using Variables

Use the following to create variables for an RF group:

- Var Block
- Adding a Varblock
- Creating Var Block Variables
- Creating Variables for RF Shape Parameters
- Var Block Calculations

Related Topics

Using RF Groups

Editing Existing RF Shapes

Var Block

A Var (variable) block contains variables and/or simulation equations. The Var block can contain absolute values or reference variables, which may contain one or more functions (Var Block Calculations).

The Var block is a member of an RF group and the variables only apply to that RF group and subgroups. The Var block symbol has no graphical representation in the layout tool.

Select the RF group in the RF Control Dialog Box, RF Design tab. The Var block popup menu enables you to:

- **Edit Var Block** — Opens the RF Shape Parameters Dialog Box
- **Cut Var Block** — Deletes the Var Block so you can paste it to another RF group.
- **Delete Var Block** — Removes the Var block from the RF group.

Related Topics

Using Variables

Creating and Modifying RF Shapes

Adding a Varblock

The varblock enables you to create variables. You can use the variables to define RF shape parameters. When you change a variable, all the RF shapes parameters with that variable change.

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Control pane** ( ).
3. Select the **RF Design** tab.
4. Select the RF Group.
5. Choose **Add Var Block** from the popup.

   The RF Shape Parameters Dialog Box appears with a default instance name (RF-<*groupname*>-varblock).

Related Topics

Using Variables	Creating and Modifying RF Shapes

Creating Var Block Variables

Define the RF shape parameters with Var block variables (Adding a Varblock).

The Equation Editor Expressions enables creation of variables and/or simulation equations. You create variables to define RF shape parameters (RF Shape Parameters Dialog Box).

Prerequisites

- The RF group must have a Var block (Adding a Varblock).

Procedure

1. Click **RF Toolkit** () to enable the RF toolbar (RF Toolkit Toolbar).
2. Click **Control pane** ( ).
3. Select the **RF Design** tab.
4. Expand the RF group that contains the Var block.
5. Select the Var block and then choose **Edit Var Block** from the popup.

   The RF Shape Parameters Dialog Box appears.
6. Type in the variable Name, Value, and select Unit in the Parameters field.
7. Click **Add**. The Select parameters field lists the new variable.
8. Add additional variables or simulation equations as needed.
9. Click **OK**.

Related Topics

Using Variables

Creating and Modifying RF Shapes

Creating Variables for RF Shape Parameters

Define RF Shape parameters with Var block variables within an RF group.

Prerequisites

- The RF shape must be a member of an RF group containing a Var Block (Adding a Varblock).
- The Var block must contain variables (Creating Var Block Variables).

Procedure

1. Click **RF Toolkit** ( ) to enable the RF toolbar (RF Toolkit Toolbar).
2. Select an RF shape.
3. Click **Parametric Properties** ( ) of the RF toolbar.
4. Select a parameter in the Select parameter field.

   The **Equation Editor** button becomes active.
5. Click **Equation Editor**.
6. Change or alter the parameter value and click **<< Insert** to insert a selected variable from the Variables field. The Variables field contains the Var block variables.

   Expressions that require evaluation must be in parentheses "()" (Var Block Calculations).
7. Click **OK**.

Results

RF Shape parameter displays the new parameter value.

Related Topics

Var Block
Replacement and Repair
Adding a Varblock

Var Block Calculations

An RF shape parameter can contain variables (expressions), which require calculation. An expression must be evaluated to create the correct value. Expressions that require evaluation must be in parentheses “()”.

The schematic capture or layout tools evaluate expressions with mathematical operators (Equation Editor Expressions). Advanced RF simulator expressions are sent to the RF simulator for evaluation.

Note

You must have an active link to the RF simulator to check the advanced RF simulator expressions (Connecting Layout to RF Simulator Server).

For example, the following parameter requires evaluation:

- L=(Lin/2+1.25*3) mm, where Lin is a variable defined by the Var instance.

Related Topics

Using Variables
Creating and Modifying RF Shapes

Create Schematic Symbol Dialog Box

To access: RF Control Dialog Box, RF Design tab, select the RF group, and choose **Generate Library Symbol**.

Description

Use this dialog box to define the symbol name and to designate the location for the symbol in the Central library. Creating a symbol for the RF shape in the central library enables you to instantiate the symbol in the schematic capture tool.

Figure 4-82. Create Schematic Symbol Dialog Box

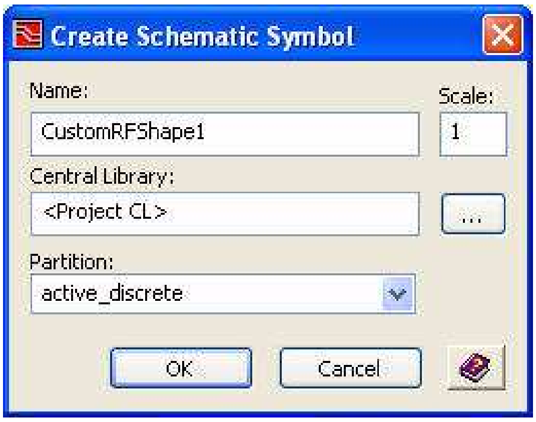


Table 4-26. Create Schematic Symbol Dialog Box Contents

Field	Description
Name	Defines the symbol name. The RF group name is the default name. If the name matches a current RF shape name, "_0" appends to the name (Example: my_group, my_group_0).
Scale	Defines the symbol display size in the schematic capture tool. The schematic symbol defaults to the display size of the RF group layout tool. The scale factor (0.2 to 20) is a multiplier of the original RF group size. In this example, the area in units has the following scaling factors: • Scale 0.2 = 10 unit • Scale 1 = 50 unit • Scale 20 = 1000 unit
Central Library	Defines the Central Library.
Partition	Defines the Central Library partition for the symbol.

Related Topics

Creating a Library Part

Editing Existing RF Shapes

Creating and Modifying RF Shapes

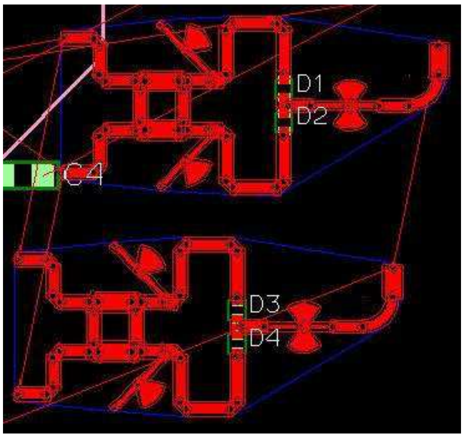
RF Shape Clearance Checking

RF Shapes from different RF groups are checked against each other using the RF-shape-to-trace clearance. Checking does not occur between RF shapes in the same RF group. Clearance checks of RF shapes use the largest RF shape-to-trace clearance value (Figure 4-83). A violation occurs when an RF shape clearance touches or overlaps a segment of another RF group (Layer-to-Layer Clearance Rules).

Regardless of RF group membership, if an RF segment violation occurs between RF groups, it is flagged as an error by DRC (Design Rule Checking) unless the RF segment has a common shape or is directly connected by a node pair (DRC RF Violations).

RF Shapes (even within the same RF group) must maintain RF shape clearance rules to design objects. The RF shape clearance is controllable in all directions including tapered clearances (Clearance Editor Dialog Box). The clearance rules are hierarchical; the RF group has the lowest priority and segments has the highest priority. The RF group clearance is the same as the RF shapes clearances within the RF group. All RF shapes within the RF group use the RF group rules (RF Group Clearance Rules Dialog Box).

Figure 4-83. RF Group to RF Group Clearance



	Mixer-1 (RF Group) Mixer-2 (RF Group)

Refer to RF Group to RF Group Clearance Check for clearance details for nested RF groups.

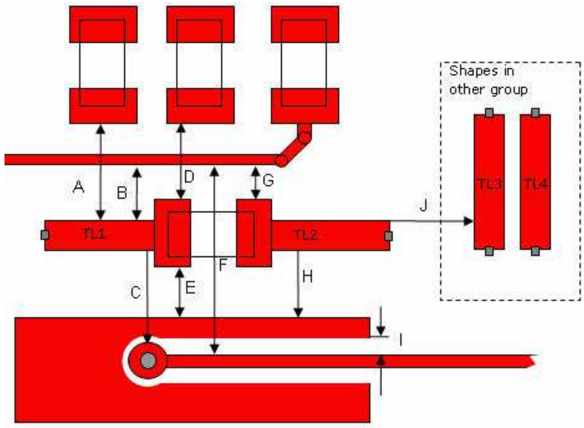
No checking occurs between RF nodes within an RF group or between RF groups. In addition no checking occurs between RF nodes and any obstructs as long as the node and the obstruct are members of the same RF group.

The following RF Group (top level) to RF Group (top level) clearances are supported:

- Trace, Via, and Pad Clearances and Connections
- RF Group to RF Group Clearance Check
- Layer-to-Layer Clearance Rules
- Overlap Check
- RF Shape Clearance Restrictions and Exceptions

Figure 4-84 gives an overview of the different rules you can set between RF shapes. Traces, vias, pads, and planes use the netclass clearances even if they belong to an RF group.

Figure 4-84. Clearance Model - Overview



A — RF shape-to-Pad	F — Trace-to-Trace (netclass)
B — RF shape-to-Trace	G — Pad-to-Trace (netclass)
C — RF shape-to-Via	H — RF shape-to-Plane
D — Pad-to-Pad (netclass)	I — Trace-to-Plane (netclass)
E — Pad-to-Plane (netclass)	J — RF shape-to-RF shape (with the trace obstruct from TL2)

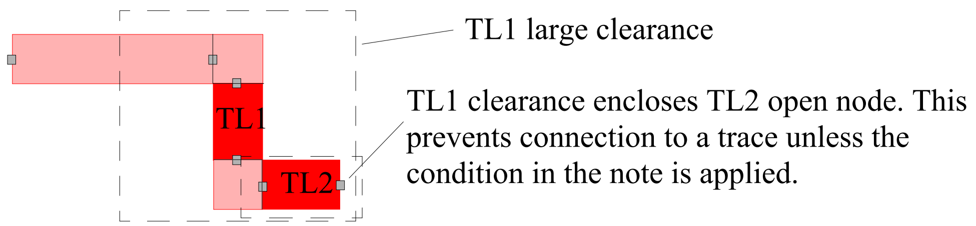
Related Topics

DRC RF Violations	Setting Clearances, Copying, and Creating Adjacent Layer Clearances
Using RF Groups	Editing Existing RF Shapes
Creating and Modifying RF Shapes	

Trace, Via, and Pad Clearances and Connections

The Clearance Editor Dialog Box enables you to set per side clearances. The clearance of one RF segment or shape can prevent the connection of a different RF segment or shape. This is illustrated in Figure 4-85. The clearance of TL1 encompasses TL2 open node, which prevents any connection.

Figure 4-85. Clearance - Route Clearances



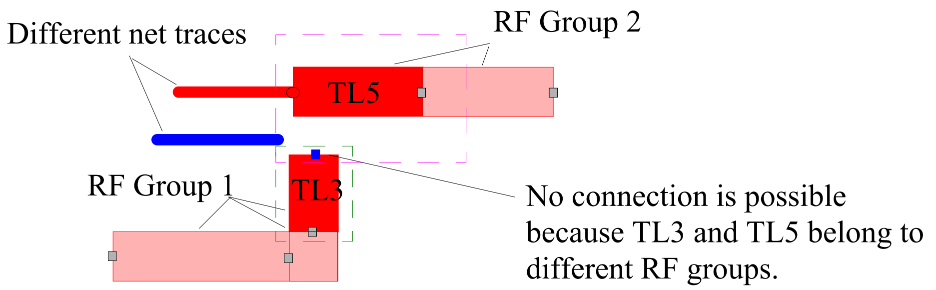
Note: A trace can connect to a node only if the trace and node belong to the same net. However, you cannot use a single node net to create a net exception.

A trace can violate multiple clearances if:

- All RF shape clearances belong to the same RF group (Figure 4-87).
- The connecting trace to a node must have the same net. Only the connecting segment can violate the RF shape clearances (RF Entry Rules Dialog Box).
- The trace clearance overlaps the node clearance.

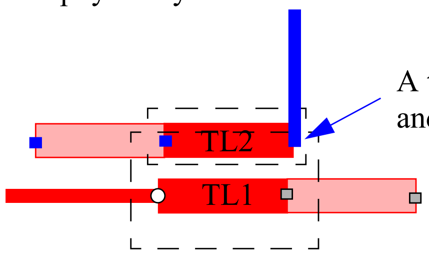
Figure 4-86 shows the open node of TL3 being overlapped by two different RF shape clearances.

Figure 4-86. Clearance - RF Shapes in Different RF Groups



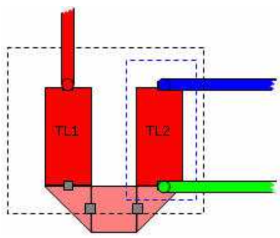
To enable a node-trace connection, the RF shapes must belong to the same RF group when the RF shape clearance overlaps a node (Figure 4-87).

Figure 4-87. Clearance - RF Shapes within an RF Group



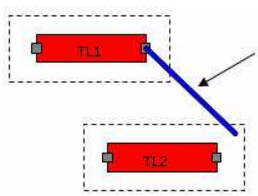
If all RF shapes belong to the same RF group, a trace is allowed to violate the RF shapes clearances and connect to a node (Figure 4-88).

Figure 4-88. Clearance - Shapes in Same Group



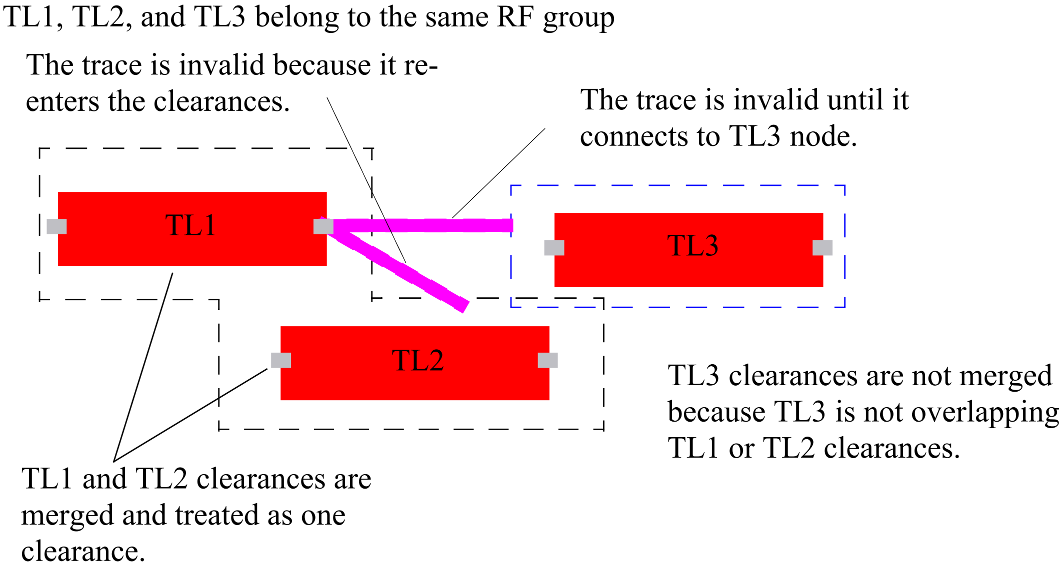
A trace segment clearing an RF shape clearance cannot enter another RF shape clearance unless it connects to the RF shape node. Figure 4-89 shows a trace violation until you connect the trace to the TL2 node.

Figure 4-89. Clearance - Trace in Violation



A trace connecting to an RF shape cannot re-enter the same RF shape clearances again. When multiple RF shape clearances overlap, they are treated as a single RF shape clearance (Trace Connection Clearance). Traces TL1 to TL2 and TL1 to TL3 in Figure 4-90 are in violation. The traces are not in violation when they connect to the nodes (TL1 to TL2 and TL1 to TL3).

Figure 4-90. Clearance - Overlapping

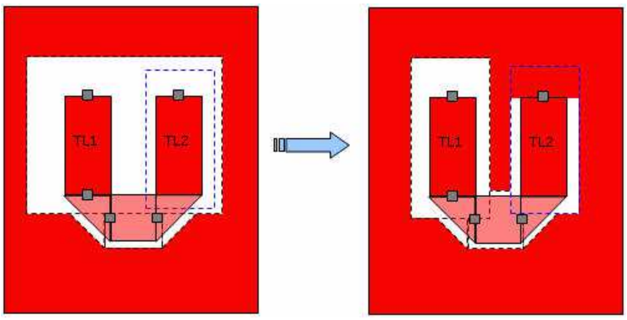


> **Note**
> The same rules apply to pad and via clearances.

A plane clearance always clears the RF shapes unless you set the RF shapes node *Connect to plane* property to **Connect** (RF Control Dialog Box). Plane shapes are not allowed to violate multiple RF shape clearances.

For example in Figure 4-91 let TL2 be an open node with a Connect-to-plane setting **Connect**. On the left side of Figure 4-91, the RF Shape-to-Plane clearance of TL1 prevents the plane connection to TL2 open node. The right side of Figure 4-91 shows a TL1 RF Shape-to-Plane clearance reduction. This enables the TL2 open node to connect to the plane (Node Properties).

Figure 4-91. Clearance - Plane Obstructs



Related Topics

DRC RF Violations

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using RF Groups
Editing Existing RF Shapes
Creating and Modifying RF Shapes
RF Shape Clearance Checking

RF Group to RF Group Clearance Check

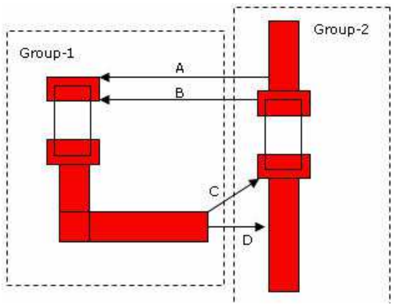
Clearance checking occurs between RF shapes and meanders when they belong to separate RF groups. Clearance checking occurs between only top-level RF groups. RF group clearance checking uses the RF groups largest RF shape to trace clearance value (RF Group Clearance Rules Dialog Box).

Note

RF Group checking only occurs between RF shapes. Other design objects are not included in this check.

Figure 4-92 shows two top-level RF groups with pads and RF shapes. A pad uses the RF-Shape-to-Pad rule against all shapes (RF Group Clearance Rules Dialog Box).

Figure 4-92. Clearance - RF Group to RF Group Clearance



A — RF Shape to pad clearance

B — Pad to pad clearance (netclass)

C — RF Shape to pad clearance

D — RF group to RF Group clearance (RF shape-to-trace value)

Related Topics

DRC RF Violations
Setting Clearances, Copying, and Creating Adjacent Layer Clearances
Using RF Groups
Editing Existing RF Shapes
Creating and Modifying RF Shapes
RF Shape Clearance Checking

Layer-to-Layer Clearance Rules

An RF shape may require clearances on adjacent layers. Create layer to layer clearance rules with the RF Group Clearance Rules Dialog Box. You create adjacent layer clearances by creating a new clearance rule and specifying the layer with the Increase/Decrease Layer Index ( ) arrows (Setting Clearances, Copying, and Creating Adjacent Layer Clearances).

Figure 4-93 shows an RF segment with layer to layer clearance rules. The physical layer of the RF shape is Index 0. Positive Index values (+1) define layers below the RF shape. Negative Index values (-1) define layers above the RF shape. The clearance rules follow the RF shape as the shape moves in the stackup (RF Shape Layer Change - Clearances).

Figure 4-93. Clearance - Layer to Layer Rules

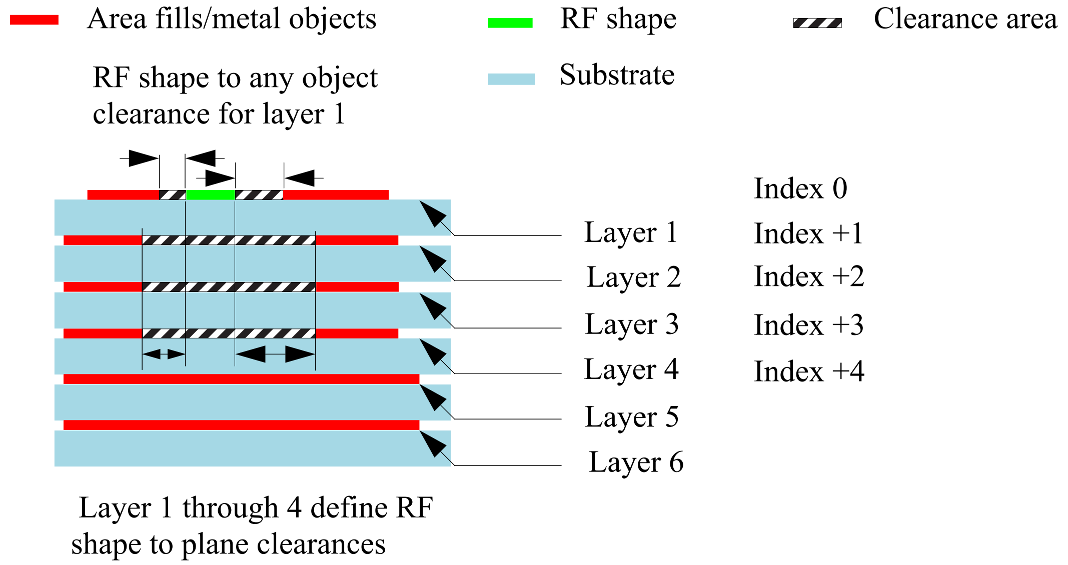
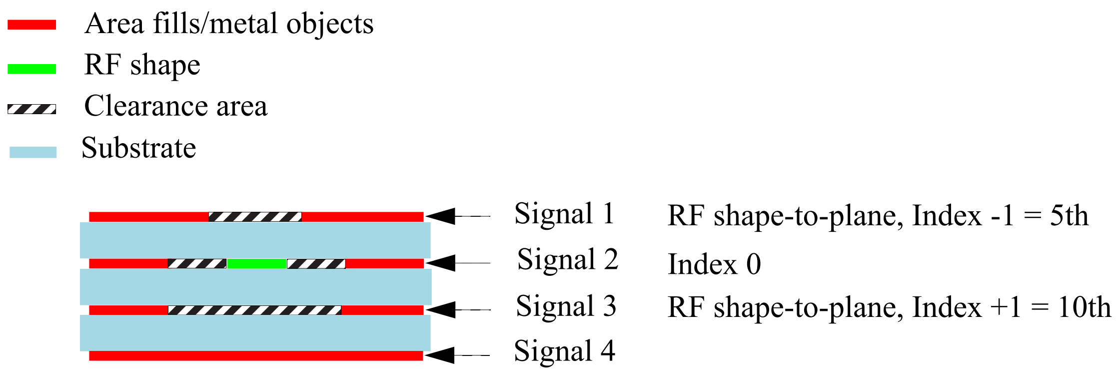


Figure 4-94 shows clearance layer rules above and below the RF shape (RF Shape Layer Change - Clearances).

Figure 4-94. Clearance - Relative Z-axis



Related Topics

DRC RF Violations

Using RF Groups

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Editing Existing RF Shapes

RF Shape Clearance Checking

RF Shape Layer Change - Clearances

The Figure 4-95 (left) shows the RF shape that has layer clearances on Signal 1 and Signal 3 (Layer-to-Layer Clearance Rules). Figure 4-95 (right) shows the results after pushing the RF shape to Signal 3. The RF shape layer clearances applying to Signal 2 and Signal 4.

Figure 4-95. Layer Clearances Shift on Push

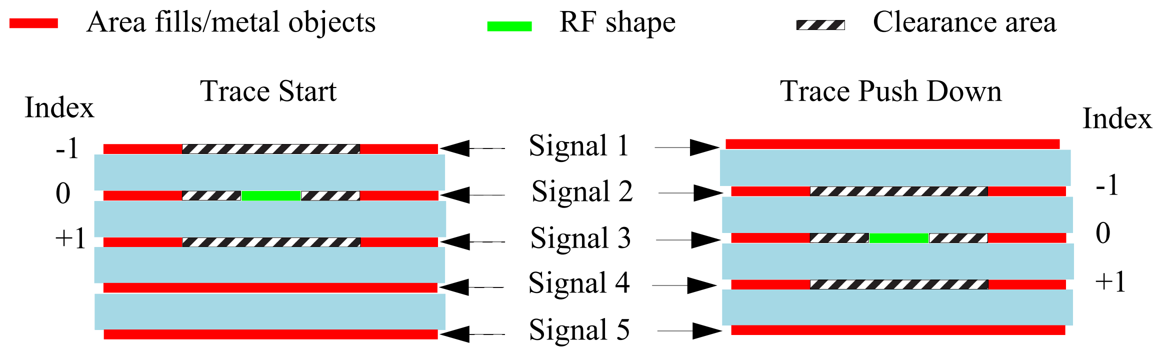
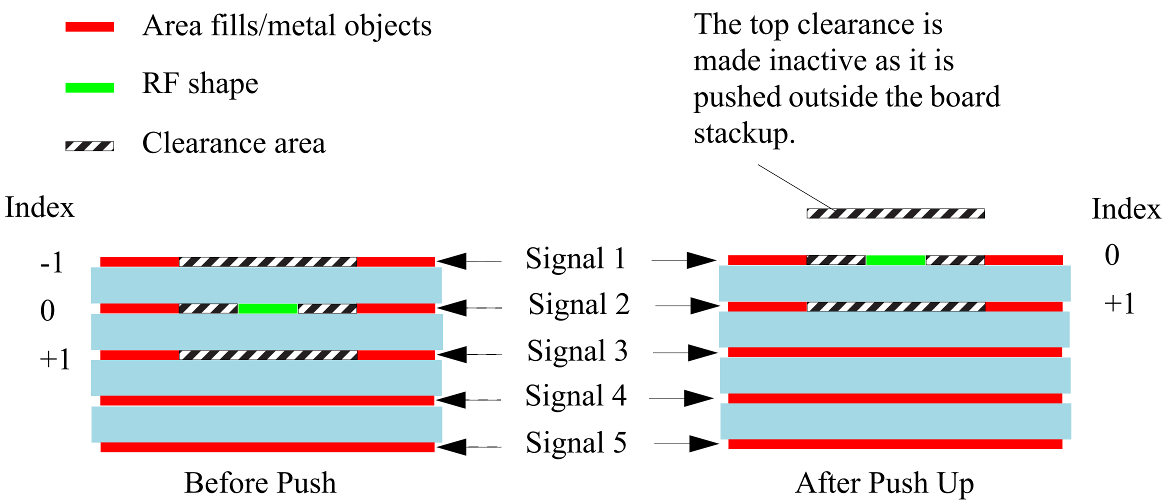


Figure 4-96 (left) shows an RF shape that has layer clearances on Signal 1 and Signal 3. Figure 4-96 (right) show the results after pushing the RF shape to Signal 1. The RF shape layer clearance only applies to Signal 2 after the push. The layer clearance above the RF shape becomes inactive. The layer clearance above the RF shape remains inactive as long as the RF shape remains on the top layer of the stackup. Pushing the RF shape down into the stackup, causes the layer clearance above the RF shape to become active. Pushing the RF shape to Signal 5, would inactivate the bottom RF shape layer clearance.

Figure 4-96. Inactive Layer Clearance



Note
CES layer and RF shape rules apply to RF shapes.

Related Topics

DRC RF Violations

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Using RF Groups

Editing Existing RF Shapes

Creating and Modifying RF Shapes

RF Shape Clearance Checking

Overlap Check

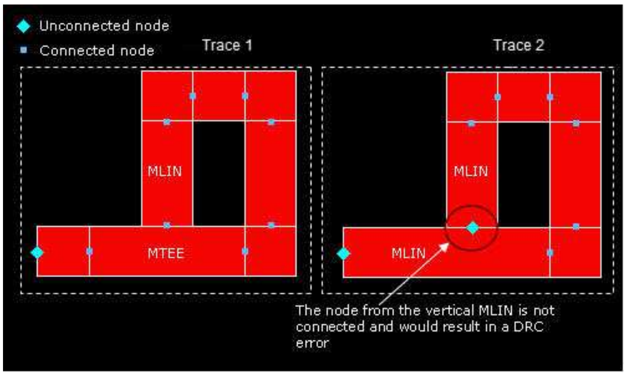
During creation of the RF circuit, DRC checking may be inhibited, which enables RF shape overlapping or edge to edge placement. Overlapping RF shapes are flagged as a DRC violation regardless of RF group membership.

Note

The non-connected node violation occurs in DRC unless you have defined the node as a no-connect node (RF Control Dialog Box).

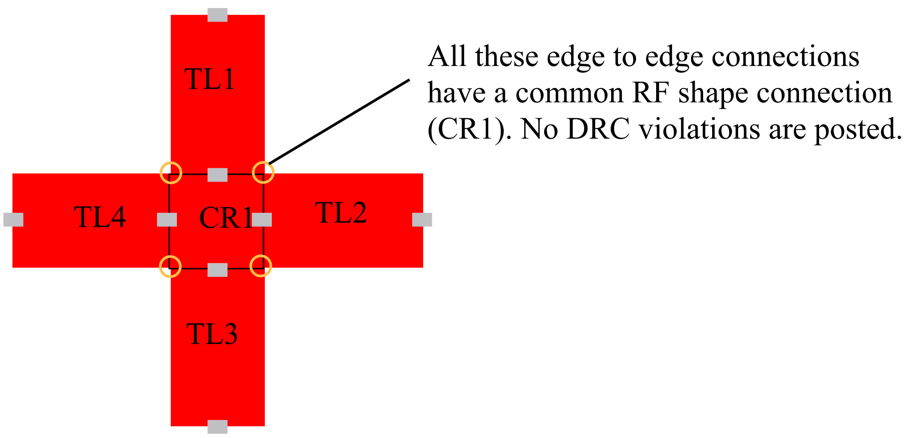
Figure 4-97 shows two separate top-level RF groups. Trace 1 RF group has RF shapes having only one open node. DRC does not post any edge to edge violations. Trace 2 RF group has an RF shape with two open nodes. DRC posts an edge to edge violation. The edge to edge violation enables you to evaluate the connection.

Figure 4-97. Clearance - Overlap Check



RF shapes not sharing a common RF shape (through connectivity) can have an edge to edge DRC violation posted. Figure 4-98 shows RF shapes sharing a common RF shape (CR1) through connectivity. No DRC violations are posted.

Figure 4-98. Overlapping Not in Error



Related Topics

DRC RF Violations

Using RF Groups

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Editing Existing RF Shapes

RF Shape Clearance Checking

RF Shape Clearance Restrictions and Exceptions

Not all RF shapes in the library support a unified clearance around the RF shape. For example, an inductor only supports RF group and shape rules.

All RF shapes supporting a per-side clearance use the Clearance corner type defined in the RF Control Dialog Box.

Related Topics

DRC RF Violations

Using RF Groups

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Editing Existing RF Shapes

RF Shape Clearance Checking

Adding and Deleting RF Elements

Add or delete RF elements from a design with the layout tool. You can also increase the Pin Spacing in the AutoPlacer dialog box (*Expedition PCB User's Guide*). This moves pin-to-pin connections for elements further apart, which enables disconnecting, deleting, adding, or connecting RF elements easier.

Note

RF port elements (those on the border of a design) **cannot** be deleted. You must set up context editing prior to deleting RF elements (*Expedition PCB User's Guide*).

Related Topics

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Replacement and Repair

Editing Existing RF Shapes

Using RF Groups

Reusing RF Schematic Capture Blocks

Reusing RF Schematic Capture Blocks

When you make changes to a replicated block, those changes are reflected in all occurrences of that block. This includes any changes you make in the layout tool to objects, nets, or their parameters.

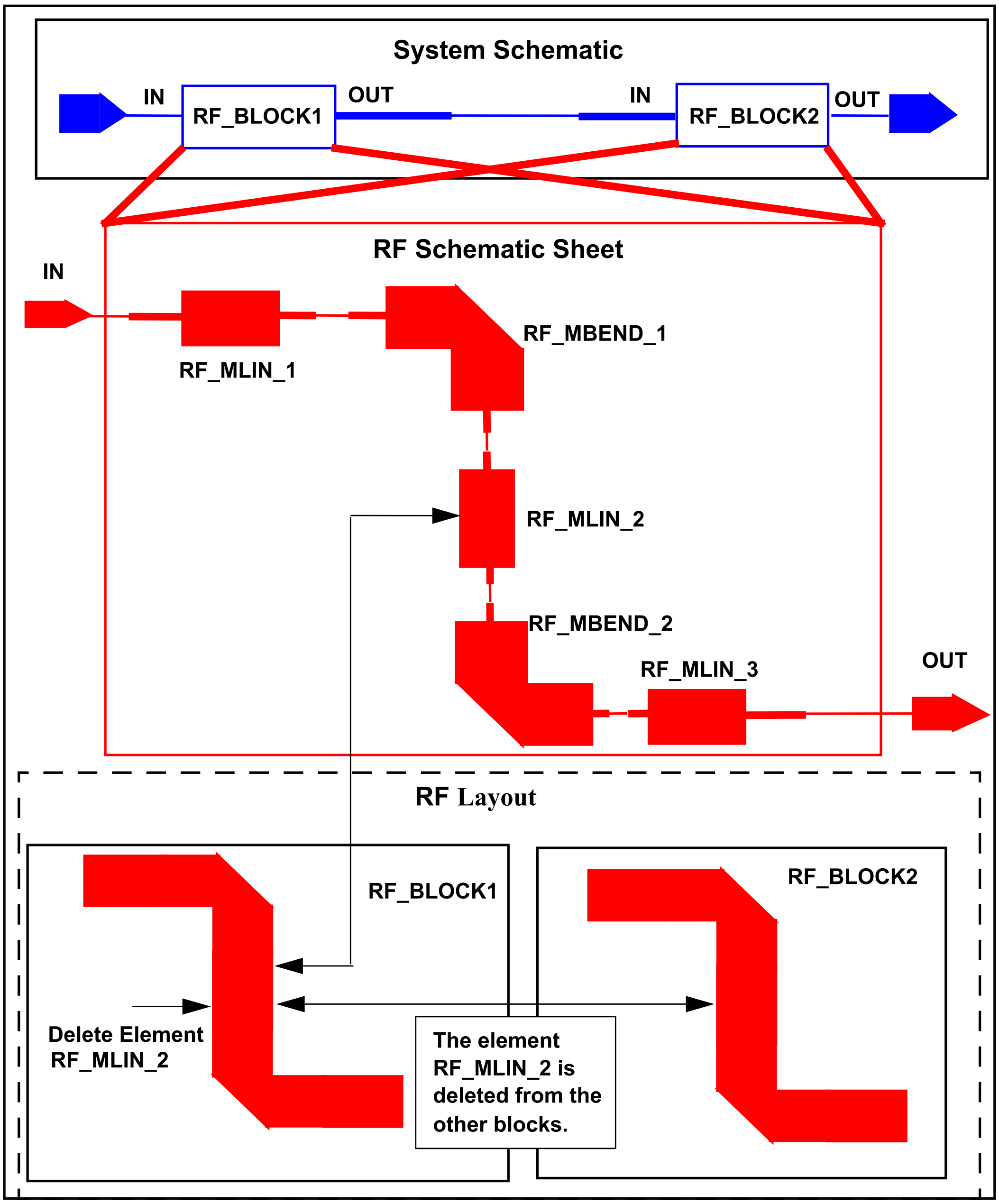
Figure 4-99 illustrates a system level schematic and the related layout representation for an RF region (block) instantiated multiple times. This provides a secure way to keep design databases synchronized, while reducing the overall work required to make changes.

When you finish your changes in layout, update the schematic sheets by synchronizing the design (Back Annotation). Synchronization modifies only one schematic capture model because the RF regions are identical and they refer to the same schematic.

Note

The process of reusing RF Blocks is *not* the same as the PCB Reuse facility.

Figure 4-99. Reuse Blocks



Related Topics

Placing RF Groups with Auto Arranger

Adding and Deleting RF Elements

Creating and Modifying RF Shapes

Setting Clearances, Copying, and Creating Adjacent Layer Clearances

Auto Arranger

Design Rules Check

Editing Existing RF Shapes

Using RF Groups

Replacement and Repair

Design Rules Check

Design Rules Check (DRC) settings control several aspects of both interactive and automatic routing in the layout tool. Design rules form the basis for checking the connections created by the router in layout.

Within an RF group or multiple RF regions, the following DRCs are ignored.

Placement DRCs for parameterized components:

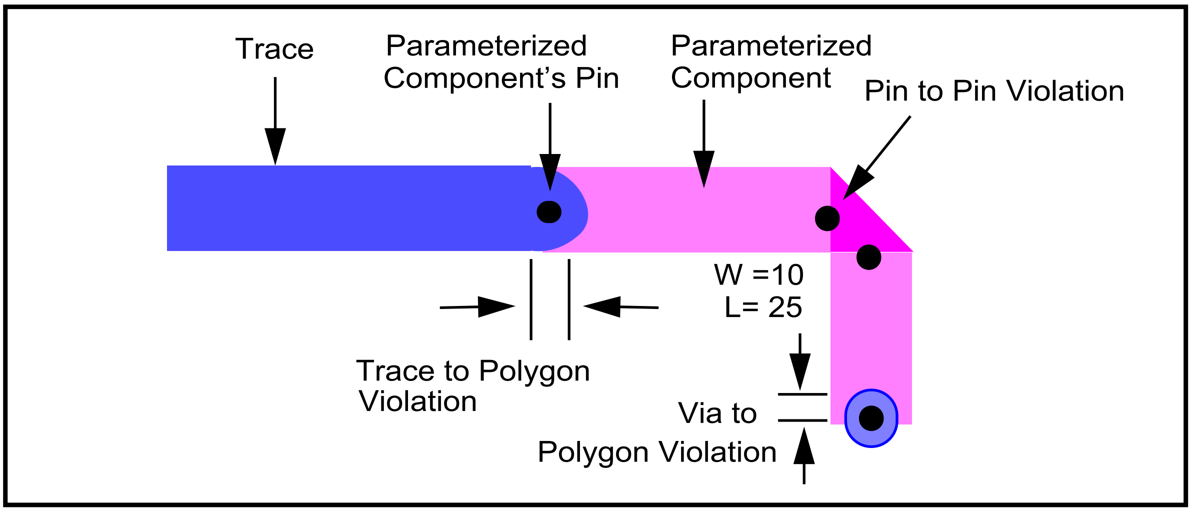
- Pin to pin.
- Component type to component type (RF Specific Component to Component Checking).

Routing DRCs:

- Trace to polygon, if the net connects to the same component pin.
- Via to polygon, if the net connects to the same pin.

DRC checking affects all components that are part of an RF group. Components that are not part of an RF group receive the full layout DRC checking for placement and routing. Figure 4-100 shows the ignored DRC checks for an RF group.

Figure 4-100. RF DRC Checks Ignored



Related Topics

RF Shape to Element Rules

RF Shape Clearance Checking

RF Specific Component to Component Checking

Distributed and discrete RF components within an RF group are checked with RF specific rules. These rules apply rather than the standard Design Rules Check (DRC).

RF components are:

- Distributed RF elements such as transmission lines, stubs, and couplers which are etched copper and are part of the PCB.
- Discrete components or packaged parts such as resistors, capacitors, and Op Amps which are placed on either the top or bottom side of the PCB.

DRC checking for RF specific rules enables the following:

- Placement of RF components on inner layers.
- Multi-layered RF components having different geometries on individual layers.
- Placement outlines that intersect provided the metal does not touch.

With DRC active, RF specific rule checking displays error messages about component violations during interactive placement. RF specific checking occurs during batch mode checking.

When you use RF Specific component-to-component checking, DRC settings are modified as follows:

- Component geometries on different layers can have polygon intersections. These violations are not considered errors.
- Two or more components can each have a node (pin) on the same net without causing a violation.
- The DRC check includes any violations between two RF distributed components and between an RF distributed component and a packaged part.

Related Topics

Design Rules Check

RF Specific Component to Component Checking

RF Shape Clearance Restrictions and Exceptions

DRC RF Violations

DRC RF Violations

RF DRC violations may occur during design development. Run **Analysis > Batch DRC** and view the following RF error and warning messages:

- RF Shape to Element Rules
- DRC RF Clearance and Overlap
- DRC Node and Pad Entry Rules
- RF Expired Nodes
- DRC RF Connectivity

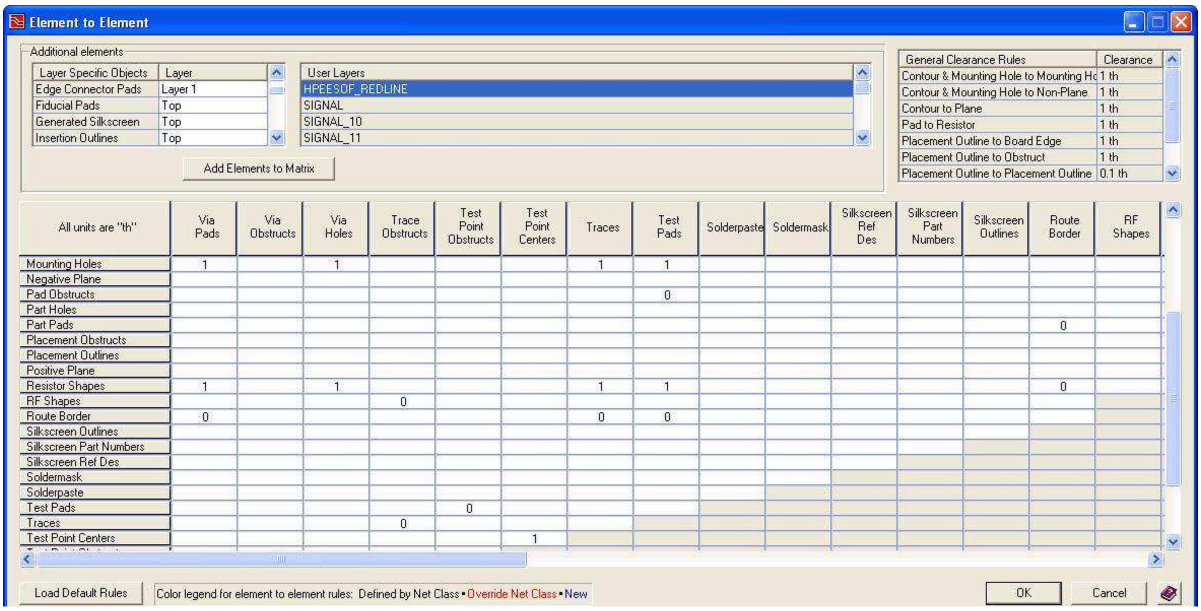
Related Topics

Design Rules Check	RF Shape Clearance Checking
Creating and Modifying RF Shapes	Editing Existing RF Shapes

RF Shape to Element Rules

Click the **Analysis > Batch DRC > Advanced Element to Element Rules** button, the Element to Element dialog box appears (Figure 4-101). The RF Shapes row defines element to element rules.

Figure 4-101. Element to Element Dialog Box



Related Topics

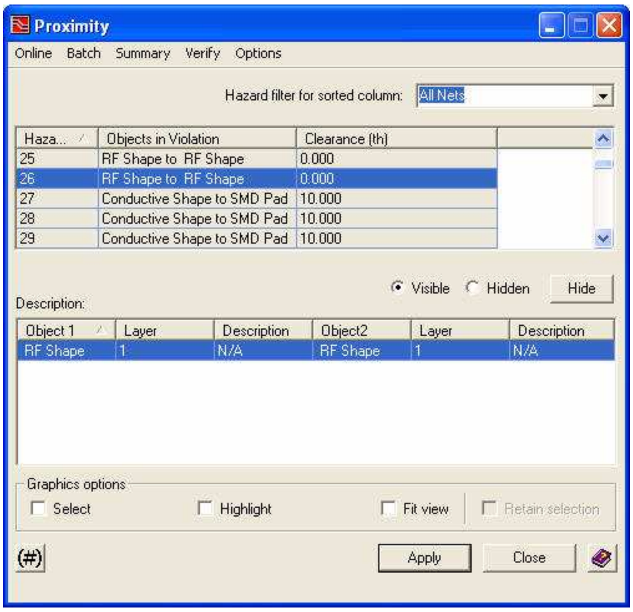
Design Rules Check	RF Shape Clearance Checking
Creating and Modifying RF Shapes	Editing Existing RF Shapes
DRC RF Violations	

DRC RF Clearance and Overlap

The RF Shape Clearance Checking violations appear in Proximity Hazards (**Analysis** > **Review Hazards** > **Batch** > **Proximity)** (Figure 4-102). The types of violations include:

- RF Groups
- RF Shape to RF Shape
- RF Shape to Trace
- RF Shape to Pad
- RF Shape to Via
- RF Shape to Plane
- Overlap Check

Figure 4-102. DRC Proximity Dialog Box



RF component violations are listed in the Components Hazard (**Analysis** > **Review Hazards** > **Online** > **Components)**.

Related Topics

Design Rules Check

Creating and Modifying RF Shapes

DRC RF Violations

RF Shape Clearance Checking

Editing Existing RF Shapes

DRC Node and Pad Entry Rules

The RF connectivity violations for node and pad entry rules are listed as Open Netlines Hazards (**Analysis** > **Review Hazards** > **Online** > **Open Netlines)**.

The RF net or partial net violations are listed as Unrouted / Partial Nets Hazards (**Analysis** > **Review Hazards** > **Batch** > **Unrouted / Partial Nets)**.

Violations to node entry rules defined in the RF Entry Rules Dialog Box are listed as Node Entry Hazards (**Analysis** > **Review Hazards** > **Online** > **Node Entry)** and have the following violation types:

- Node-to-trace entry angle violation. The trace entry/exit is in violation (RF Shape Node Entry Angle).
- Node-to-trace escape violation. The trace changed direction before it cleared the shape (Trace Connection Clearance).
- No connection from Node-to-trace. The trace is not truly connected to the node (Node-to-Node and Node-to-Trace Rules).
- Node-to-trace unknown violation. None of the above violation types apply.

Related Topics

Design Rules Check

Creating and Modifying RF Shapes

DRC RF Violations

RF Shape Clearance Checking

Editing Existing RF Shapes

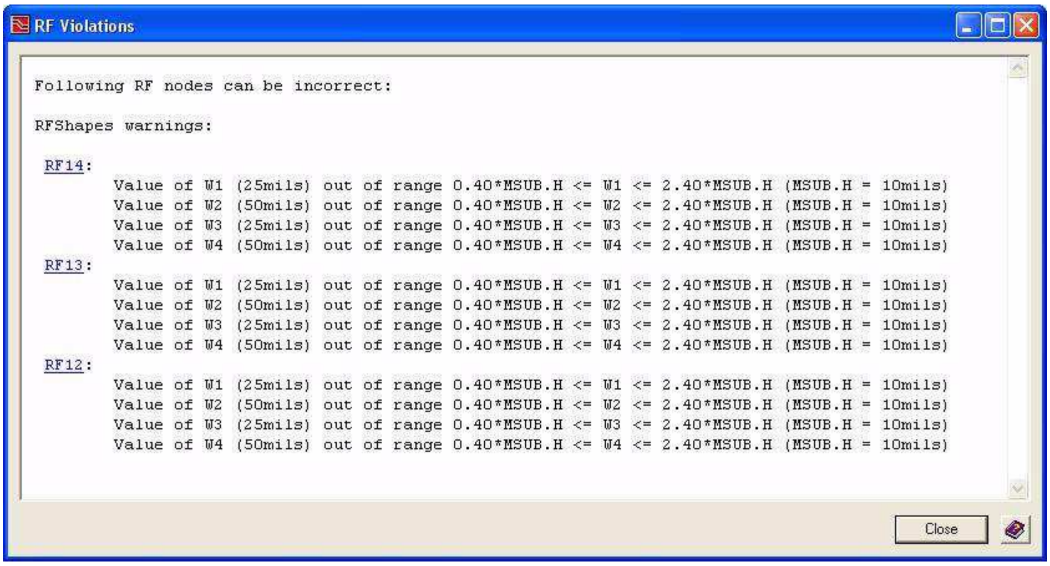
RF Expired Nodes

When updating RF shapes, the RF design nodes in violation are written in the RF Violation ASCII file (**Analysis > Review Hazards > Batch > RF Violations).**

The file lists all floating nodes in violation (Figure 4-103). You must manually change the Expired property to **True** in the RF Control Dialog Box, RF Design tab, Node Properties field, to remove the floating node violation.

Additional violation information appears in the RF Violations file.

Figure 4-103. RF Violations Display



Related Topics

Design Rules Check

Creating and Modifying RF Shapes

DRC RF Violations

RF Shape Clearance Checking

Editing Existing RF Shapes

DRC RF Connectivity

RF design connectivity errors are reported in **Analysis** > **Review Hazards** > **Online** > **Open Netlines** and **Analysis** > **Review Hazards** > **Batch** > **Unrouted / Partial Nets**.

Related Topics

Design Rules Check

RF Shape Clearance Checking

Creating and Modifying RF Shapes

Editing Existing RF Shapes

DRC RF Violations

Chapter 5
Generating Manufacturing Data

Post Processing an RF Design

Before you generate manufacturing data for a PCB design that contain RF circuits, you must make sure the data is precise. Typically, RF circuit dimensions are much smaller that those of analog or digital designs and require a more accurate format.

Make sure to review hazards before generating output. For more information on reviewing hazards, refer to the *PCB Verification Guide*.

For more information on generating manufacturing outputs, refer to *Manufacturing Inputs and Outputs Guide*.

Note

The RF-specific DRC checks and the clearance values that you set in PCB Layout are honored in FabLink XE Pro. These values are only viewable in FabLink XE Pro.

Related Topics

RF Layout

Chapter 6
Importing a Design

You can import your design from the Advanced Design System (ADS) RF simulator into the schematic capture tool in the IFF (Intermediate File Format). Schematic RF elements created in non-Mentor Graphics schematic capture tools are not imported. RF shapes are added within the schematic capture tool and meanders are added in layout.

Importing ADS Data into Layout

You can update placement and parameter settings in ADS and import them into the layout tool.

Meander lines (traces) and floating metal are imported from ADS into the layout design as area fills. Figure 6-1 shows the ADS layout toolbar that you use to create objects and text to be imported. All of these previously listed elements (except polylines) import as floating metal, which are area fills not connected to a net. The imported area fills are associated with an RF group which you can rotate and move with the components in the RF group.

Figure 6-1. ADS Layout Toolbar

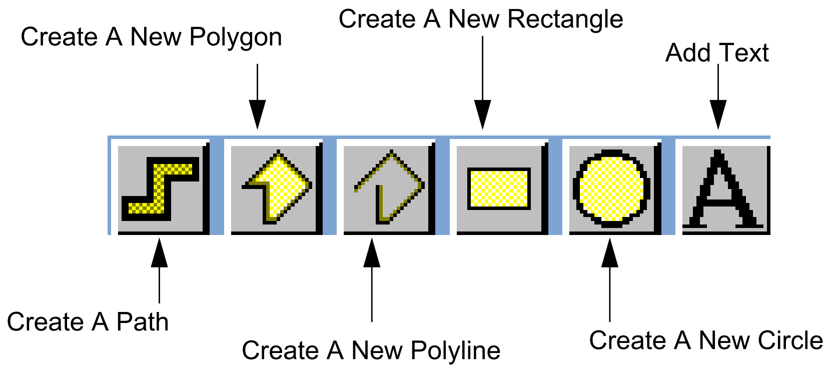
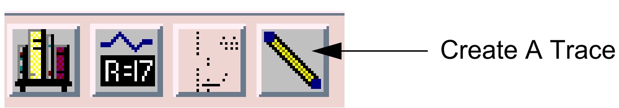


Figure 6-2 shows the ADS **Create A Trace** icon. It creates meander lines, which connect component pins to existing meander lines. You import these ADS traces into the layout design as meander lines. You can also create an ADS trace that starts and stops on another trace and is not supported in the layout tool.

Figure 6-2. ADS Create Traces Icon Layout Toolbar



Use the following procedure to import a RF simulator design into layout.

Prerequisites

There is a design in the RF simulator. (Refer to RF simulator documentation).

Procedure

1. (Optional) enter an area fill aperture size in board units.

   The aperture size affects floating meander lines and meander traces. If you do not specify a value, the software assignes a small aperture value to attempt to make the actual shape close to the input shape of the area fill. The default values are: 0.001 mm, 0.05 mils, and 0.00005 inches.

2. Create an IFF file.

3. Click **OK** to import the IFF data into the Mentor Graphics layout tool.

   When the import process is finished, all modified parameters are updated. The new shapes appear on the board, and the correct placement is updated.

   During import, the layout, X/Y location, orientations, and RF instance parameters (including hierarchical parameters) are read in for all instances. These instances map to equivalent instances already present in the Mentor Graphics design. The software generates a warning message for any elements that cannot be mapped to an equivalent element in the Mentor Graphics layout tool and maintains RF region information, such as rotation and parameter changes, for successive imports. For connectivity changes, you must synchronize with and translate from the schematic.

Results

The RF simulator design displays in the layout tool.

Related Topics

IFF Import Dialog Box

Chapter 7
Exporting an RF Design

Use the following to export schematic capture and layout design data IFF (Intermediate File Format) files.

- Exporting an RF Design from Schematic Capture
- Exporting an RF Design from Layout

Related Topics

Importing a Design

RF Layout

Exporting an RF Design from Schematic Capture

Agilent IFF syntax requires you to specify variable definitions in the schematic *.iff* files only. Any changes to variables (including the addition of new variables) requires a schematic IFF transfer of Agilent EEsof data followed by a transfer of layout IFF data.

Use the following procedure to create an IFF file from the schematic capture tool.

Prerequisites

- A schematic must contain an RF group (Creating an RF Group).

Procedure

1. Select the RF Groups tab in the Navigator window.
2. Select an RF group for exporting.
3. Choose **Generate Schematic Data**.

   The software generates an *.iff* file at:

   <*project directory*>\RF\netlist\iff\<*project name*>_<*group name*>_<*current date* (yyyymmdd)>_<*current time* (hhmmss)>*.iff*

Results

The project directory contains the *.iff* file with the correct time stamp.

Related Topics

Exporting an RF Design from Layout | Exporting an RF Design

Exporting an RF Design from Layout

When you export a design you can export the entire design or only a selected area. All components on the board that are part of the RF group are included in the export. The resulting *.iff* file includes information such as locations, orientations, and RF instance parameters (including hierarchical paramaters).

Note

Agilent IFF syntax requires you to specify variable definitions in the schematic *.iff* files only. Any changes to variables (including the addition of new variables) requires a schematic IFF transfer of Agilent EEsof data followed by a transfer of layout IFF data.

Use the following procedure to export an RF design from the layout tool to an RF simulator.

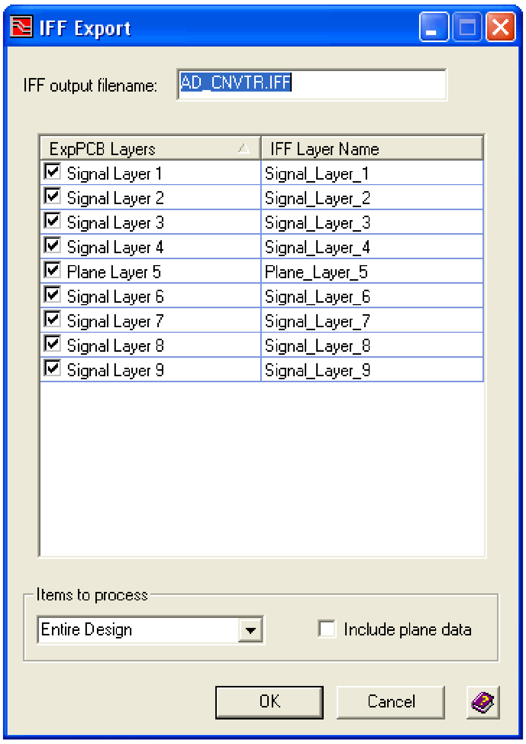
Prerequisites

- The layout tool must contain an RF design.
- The schematic capture and layout tools must be synchronized. (The design data has been forward and back annotated.)

Procedure

1. Select the area to simulate in the layout tool.
2. Open the IFF Export Dialog Box (**File > Export > IFF**) - Figure 7-1.

Figure 7-1. IFF Export Dialog Box



3. Type the name for the IFF file.
4. Select layers, areas and plane data as needed.

Caution

When you export IFF files, the existing IFF files are overwritten.

5. Click **OK**.

   The software outputs design data to the file *<design_name>\pcb\EE\Output\<filename>/.iff*.

Results

The contents of the *.iff* files contain the selected layout design objects.

Chapter 8
RF Simulator

RF Simulator Tool

The RF simulator provides a server to which the schematic capture and layout tools connect. This connection acts as a dynamic-link, which enables bidirectional data transfers.

Schematic capture and layout tools use the RF simulator to simulate RF groups. RF groups are sent to the RF simulator over the dynamic-link. The portion of the RF group sent to the RF simulator with the layout tool is described in RF Group Elements Exported to RF Simulator.

The following procedures apply to the schematic capture and layout tools after the RF simulator server is active (Launching the ADS Server or Launching MWO Server):

- Connecting Schematic Capture to RF Simulator Server
- Connecting Layout to RF Simulator Server
- RF Group Elements Exported to RF Simulator
- Post Back Hierarchical Symbols

Related Topics

RF Layout

RF Simulator

MWO RF Simulator

RF Connect Dialog Box (Schematic Capture)

To access: With the RF Toolbar active, click **RF Connect**

Description

Use this dialog box to connect the schematic capture tool to the RF simulator server.

When you connect the schematic capture tool to the RF simulator server, you create a dynamic-link, which enables bidirectional data transfers.

The schematic capture RF Connect dialog box has the following tabs:

- RF Connect Dialog Box (Schematic Capture) - Connections Tab — use this tab to connect to the RF simulator server.
- RF Connect Dialog Box (Schematic Capture) - Configuration Tab — use this tab to define the layout connection information.

RF Connect Dialog Box (Schematic Capture) - Connections Tab

To access: With the RF Toolbar active, click **RF Connect**  , select the **Connections** tab

Description

Use the Connections tab to connect (or disconnect) the schematic capture tool and the RF simulator server.

Figure 8-1. RF Connect Dialog Box (Schematic Capture) - Connections Tab

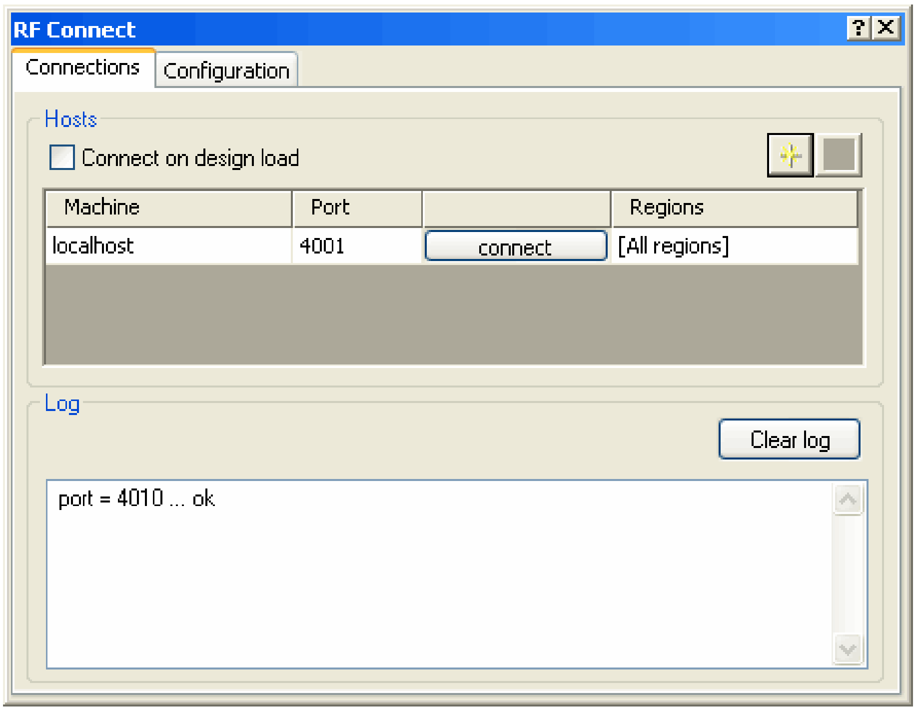


Table 8-1. RF Connect Dialog (Schematic) - Connections Tab Contents

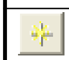
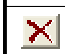
Field	Description
Connect on design load	Checked, automatically creates a connection to the RF simulator server when you load a design in the schematic capture tool. Unchecked, does not automatically connect the schematic capture tool to the RF simulator server (Connecting Schematic Capture to RF Simulator Server).
	Populates the table with active RF simulator server information. This also enables you to connect to or disconnect from the RF simulator server via the dynamic link.
	Deletes the selected RF simulator server from the list.
Machine	Defines the machine name or IP address of the RF simulator server.

Table 8-1. RF Connect Dialog (Schematic) - Connections Tab Contents (cont.)

Field	Description
Port	Defines the connection port of the machine.
connect/ disconnect	Connects to or disconnects from the RF simulator server.
Regions	Defines the RF simulator workstation location to which the RF data is sent. If you have several RF servers, each can simulate a different RF group.
Clear log	Clears the log field.
Log field	Displays a connection transcript.

Related Topics

RF Simulator Tool

RF Connect Dialog Box (Schematic Capture)

RF Connect Dialog Box (Schematic Capture) - Configuration Tab

To access: With the RF Toolbar active, click **RF Connect** , select the **Configuration** tab

Description

Use the configuration tab to define the schematic capture tool connection information (Figure 8-2).

Figure 8-2. RF Connect Dialog Box (Schematic Capture) - Configuration Tab

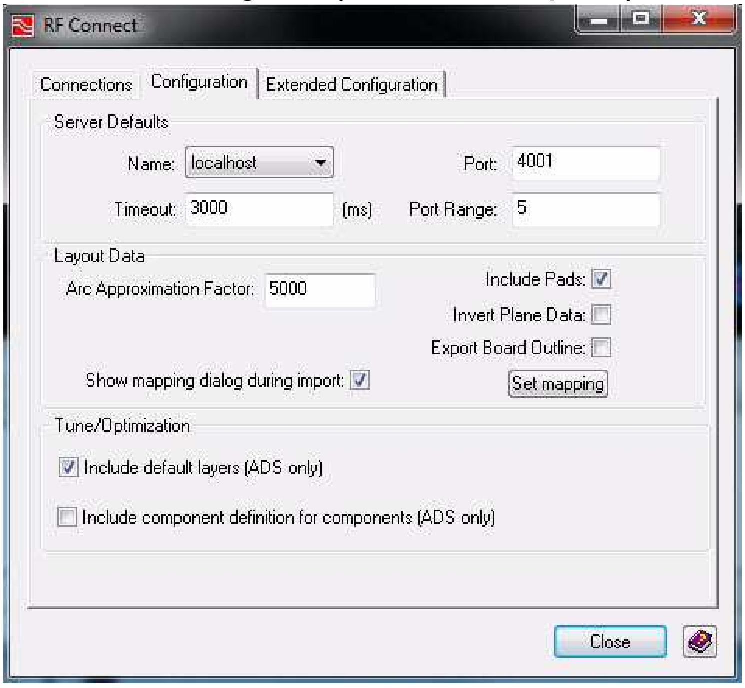


Table 8-2. RF Connect Dialog (Schematic) - Configuration Tab Contents

Field	Description
Server Defaults section — Defines the default values of the RF simulator server selected on the Connections tab.	
Name	Defines the machine name or IP address of the RF simulator server.
Timeout	Defines the maximum time allotted to connect to the RF simulator server.
Port	Defines the port address of the RF simulator server.
Port Range	Defines the number of alternate ports.
Tune/Optimization section — Determines the tune/optimization data that is imported to ADS.	

Table 8-2. RF Connect Dialog (Schematic) - Configuration Tab Contents

Field	Description
Include default layers (ADS only)	Specifies that ADS uses the default layer stackup from ADS for tuning/optimization. If you do not specify this option, ADS uses only the layer information exported from Expedition to tune/optimize.
Include component definition for components (ADS only)	Specifies that tune/optimization information for components (pin position, geometry, and so on) is included in the data imported to ADS. This allows ADS to create compatible library parts. This option is useful if there is no compatible ADS design kit available for packaged parts. If you do not specify this option, component information from Expedition is not included in the tune/optimization data that is imported to ADS.

Related Topics

Connecting Schematic Capture to RF Simulator Server

RF Connect Dialog Box (Schematic Capture)

RF Simulator Tool

Connecting Schematic Capture to RF Simulator Server

Connect the schematic capture tool and the RF simulator server with the RF Connect dialog box. The connection creates a dynamic-link, which enables bidirectional data transfers.

Note
The schematic capture and layout tools can connect simultaneously to the same RF simulator server.

Prerequisites

- The RF simulator server must be active (Launching the ADS Server or Launching MWO Server).
- The schematic capture tool displays the RF Toolbar.

Procedure

1. Open the RF Connect Dialog Box (Schematic Capture) (click **RF Connect** on the RF Toolbar) and select the Connections tab.
2. Click to populate the table with all the available RF simulator servers.

   The Configuration tab default values correspond to the RF server default values.
3. Click **connect** to connect the schematic capture tool to the RF simulator.

Results

The Log field displays an **ok**, indicating that the connection has been made successfully.

Related Topics

RF Simulator Tool

Getting Started

RF Connect Dialog Box (Schematic Capture)

RF Connect Dialog Box (Layout)

To access: With the RF Toolkit ( ) active, click **RF Connect** on the RF Toolkit Toolbar.

Description

Use this dialog box to connect the layout tool to the RF simulator server. When you connect the layout tool to the RF simulator server, you create a dynamic-link, which enables bidirectional data transfers.

The RF connect dialog box has the following tabs:

- RF Connect Dialog Box (Layout) - Connections Tab — use this tab to connect to the RF simulator server.
- RF Connect Dialog Box (Layout) - Configuration Tab — use this tab to define the RF simulator server and layout configuration information.
- RF Connect Dialog Box (Layout) - Extended Configuration Tab — use this tab to define a specific portion of an RF group to be sent to the RF simulator.

Related Topics

Connecting Layout to RF Simulator Server   RF Simulator Tool

RF Connect Dialog Box (Layout) - Connections Tab

To access: With the RF Toolkit ( ) active, click **RF Connect** on the RF Toolkit Toolbar, select the **Connections** tab.

Description

Use this tab to connect (or disconnect) the layout tool and the RF simulator server.

Figure 8-3. RF Connect Dialog Box (Layout) - Connections Tab

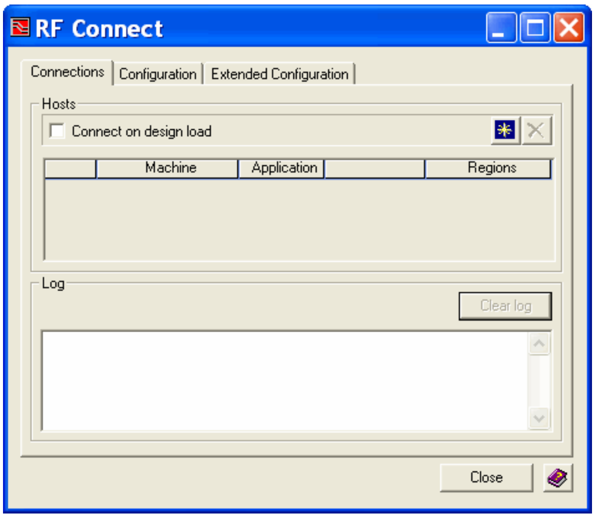


Table 8-3. RF Connect Dialog Box (Layout) - Connections Tab Contents

Field	Description
Connect on design load	Checked, creates an automatic connection to the RF simulator server when you load a design in the layout tool. Unchecked, does not create an automatic connection between the layout tool and the RF simulator server (Connecting Layout to RF Simulator Server).
Add connection	Populates the table with active RF simulator server information. This also enables you to connect to or disconnect from the RF simulator server via the dynamic link.
Delete	Deletes the selected RF simulator server.
Machine	Defines the machine name or IP address of the RF simulator server.
Port	Defines the connection port of the machine.
connect/ disconnect	Connects to or disconnects from the RF simulator server.

Table 8-3. RF Connect Dialog Box (Layout) - Connections Tab Contents

Field	Description
Regions	Defines the RF simulator workstation location to which the RF data is sent. If you have several RF servers, each can simulate a different RF group.
Clear log	Clears the log field.
Log field	Displays a connection transcript.

Related Topics

Connecting Layout to RF Simulator Server

RF Simulator Tool

RF Connect Dialog Box (Layout)

RF Connect Dialog Box (Layout) - Configuration Tab

To access: With the RF Toolkit ( ) active, click **RF Connect** on the RF Toolkit Toolbar, select the **Configuration** tab.

Description

Use this tab to define the RF simulator server and layout tool connections and create a mapping file for the ADS RF simulator (RF Mapping rflayermap.xml Dialog Box).

Figure 8-4. RF Connect (Layout) - Configuration Tab

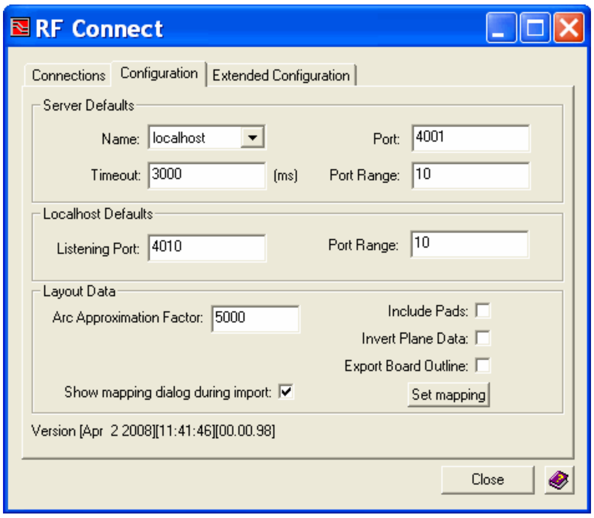
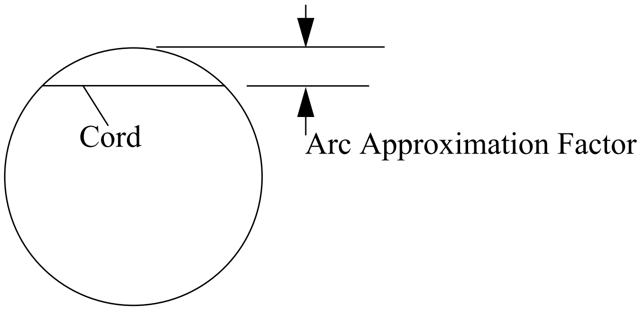


Table 8-4. RF Connect Dialog Box (Layout) - Configuration Tab Contents

Field	Description
Server Defaults section — Defines the default values of the RF simulator server selected on the Connections tab.	
Name	Defines the machine name or IP address of the RF simulator server.
Timeout	Defines the maximum time allotted to connect to the RF simulator server.
Port	Defines the port address of the RF simulator server (matches the port number of the RF simulator server).
Port Range	Defines the number of alternate ports.
Localhost Defaults section — Defines the default connection values for the layout tool.	
Listening Port	Defines the listening port address.
Port Range	Defines the port address range.

Table 8-4. RF Connect Dialog Box (Layout) - Configuration Tab Contents

Field	Description
Layout Data section — Defines the design data being transferred.	
Arc Approximation Factor	Defines the level of segmentation in the RF simulator in terms of the tolerated distance from the arc to the cord. Used by the RF simulator to control the accuracy of arcs in the data. 
Include Pads	Checked, sends non-RF part data to the RF simulator.
Invert Plane Data	Checked, sends negative plane data to the RF simulator.
Export Board Outline	Checked, sends board outline data to the RF simulator.
Show mapping dialog during import	Checked, opens the RF Mapping rflayermap.xml Dialog Box when you import an RF simulator design or RF group.
Set mapping	Opens the RF Mapping rflayermap.xml Dialog Box, which enables you to define the ADS RF simulator notations for layers, pins (Ports - ADS), and vias that are mapped to the layout tool.

Related Topics

Connecting Layout to RF Simulator Server

RF Simulator Tool

RF Connect Dialog Box (Layout)

RF Connect Dialog Box (Layout) - Extended Configuration Tab

To access: With the RF Toolkit ( ) active, click **RF Connect** on the RF Toolkit Toolbar, select the **Extended Configuration** tab.

Description

Use this tab to define the portion of the RF Group that is transmitted to the ADS RF simulator.

Figure 8-5. RF Connect Dialog Box (Layout) - Extended Configuration Tab

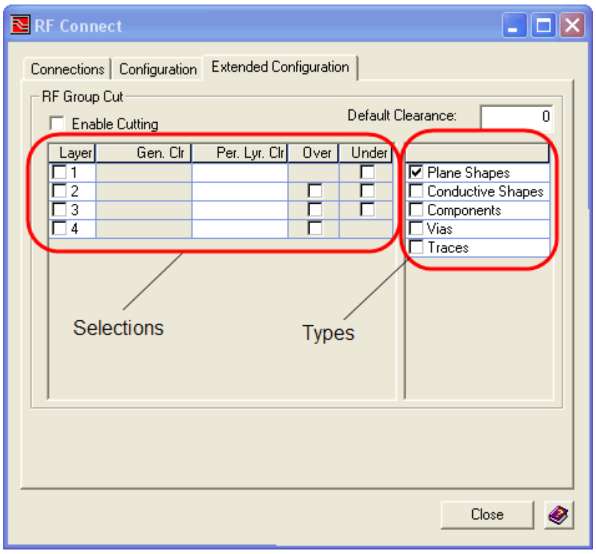


Table 8-5. RF Connect Dialog Box (Layout) - Extended Configuration Tab Contents

Field	Description
RF Group Cut section — Defines the information that is sent to the ADS RF simulator.	
Enable Cutting	Checked, restricts the RF group simulation to only those items that are selected in the Selections table and Types table. Unchecked, sends the entire RF group data to the RF simulator.
Default Clearance	Defines the default clearance values for all RF Types sent to the RF simulator. You can override the default clearance for a particular layer by specifying a per layer clearance in the Selections table (Using Extended Configuration, Enable Cutting and Default Clearance).
Selections section — Defines the options that are exported to the RF simulator.	
Layer	Checked, exports the layer to the RF simulator.
General Clearance	Defines the oversize value used to expand the RF group size. This overrides the Default Clearance value.

Table 8-5. RF Connect Dialog Box (Layout) - Extended Configuration Tab Contents (cont.)

Field	Description
Per Layer Clearance	Defines the oversize value used on the layer to expand the RF group size. This overrides the Default Clearance and General Clearance values.
Over	Checked, applies the clearance values to layers above the selected layer.
Under	Checked, applies the clearance values to layers below the selected layer.
Types	Checked, exports the RF group items to the RF simulator.

Related Topics

Connecting Layout to RF Simulator Server   RF Simulator Tool

RF Connect Dialog Box (Layout)

Connecting Layout to RF Simulator Server

Connect the layout tool and the RF simulator server with the RF Connect Dialog Box (Layout). The connection creates a dynamic-link, which enables bidirectional data transfers.

Prerequisites

- The RF simulator server is active (Launching the ADS Server or Launching MWO Server).
- The RF Toolkit ( ) is active in the layout tool (Opening the RF Toolkit).

Procedure

1. Open the RF Connect Dialog Box (Layout) (Click RF Connect), and select the **Connections** tab.
2. Click to populate the table. The Machine column displays the localhost. (The RF simulator and the Mentor Graphics tool can reside on the same workstation.)

Note
If the RF simulator server is not listed, type the name or IP address of the remote server in the Machine column.

3. Click **connect**.

Note
If no connection is made through the port or IP address, check your firewall.

Results

The Log field displays an **ok**, indicating that the connection has been made successfully.

Related Topics

RF Simulator Tool

RF Connect Dialog Box (Layout)

RF Group Elements Exported to RF Simulator

From the layout tool, you can export selected RF groups to the RF simulator. Non-RF traces and vias are exported only if they are:

- Directly connected to an RF shape or meander.
- Indirectly connected through one or more (5 pin max) device(s) that are connected to an RF shape or meander.

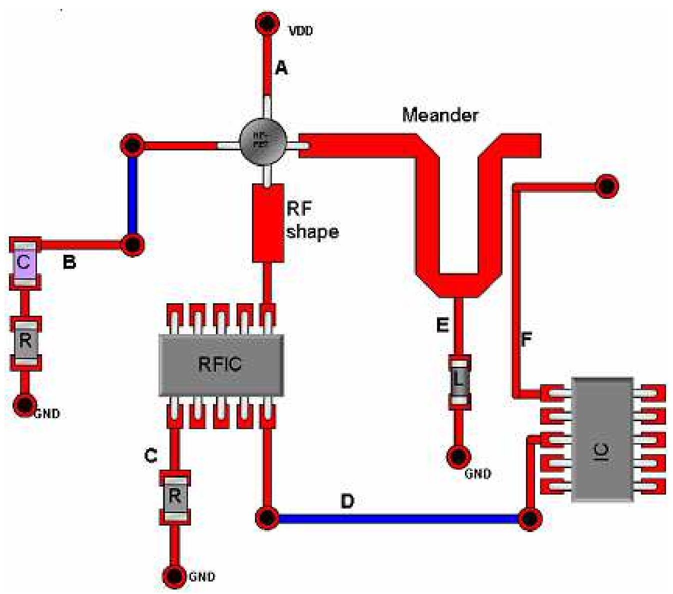
If non-RF traces and vias are required for analysis, you can use Extended RF Simulation. You can also export non-RF traces and vias to the RF simulator with an *.iff* file (Exporting an RF Design from Layout).

For example, Figure 8-6 shows an RF Group that was exported from the layout tool to the RF simulator with either the **Send netlist**, **Send layout data**, or **Send Tune/Opt Data** menu selections (RF Group Menu Selections).

The following RF and non-RF parts are sent to the RF simulator.

- Trace connections A, B, and E are exported because they are directly or indirectly connected to RF shapes or meanders.
- Trace connections C, D, and F are **not** exported because they do not meet the requirements. An error message displays listing the removed traces or vias.

Figure 8-6. Trace Via Export



Related Topics

Connecting Layout to RF Simulator Server RF Simulator Tool

Post Back Hierarchical Symbols

When you create RF designs in the RF simulator, you can use an electrical equivalent of a simulation equivalent circuit for passive or active parts. The underlying circuit represents package parameters, non-linear models, and other simulation items. This circuit has no physical meaning, so the hierarchical symbol for the part needs to be updated for the packaged part.

To import such a design into the DxDesigner-Expedition flow from the RF simulator, modify the Central Library (CL) symbol file. Edit the hierarchical symbol file before you import the IFF file. The CL hierarchical symbol name must be the same as the RF simulator symbol file. A hierarchical symbol has the Symbol Type set to **Composite** and ForwardPCB set to **True**.

Related Topics

Connecting Layout to RF Simulator Server RF Simulator Tool

Chapter 9
ADS RF Simulator

Note

Linux - On a Linux platform, in order to use the ADS RF simulator, you must add the following at the beginning of the ADS start up script:

```
LD_LIBRARY_PATH=${SDD_HOME}/common/linux/lib:${LD_LIBRARY_PATH}
```

The script is located at $HPEESOF_DIR/bin/ads.

The schematic capture and layout tools use the RF simulator Advanced Design System (ADS) to simulate RF groups.

When you create a design in the schematic capture tool and send the schematic data to ADS, all port numbers are set to 0 (Num=0). You must incrementally renumber all of the ports, starting at 1 (Num=1) in order to simulate the RF group.

The software preserves ADS port numbers when you send the design to the schematic capture tool. When you use the schematic capture tool to create additional ports for a simulated RF group, you must renumber the new ports.

The following sections address specific ADS functionality:

- Adding ADS RF Shapes to Central Library
- Layout Tool and ADS Workflow
- Installing the MentorDA_DK Design Kit
- Launching the ADS Server
- Connecting Schematic Capture to RF Simulator Server
- Send Design Data to ADS Simulator
- Generate Data
- Sending ADS Design to Schematic Capture Tool
- Sending ADS RF Design to Layout Tool
- RF Mapping rflayermap.xml Dialog Box
- Creating an ADS Mapping File

- Using Extended Configuration, Enable Cutting and Default Clearance
- Post Back to Schematic Capture and Layout

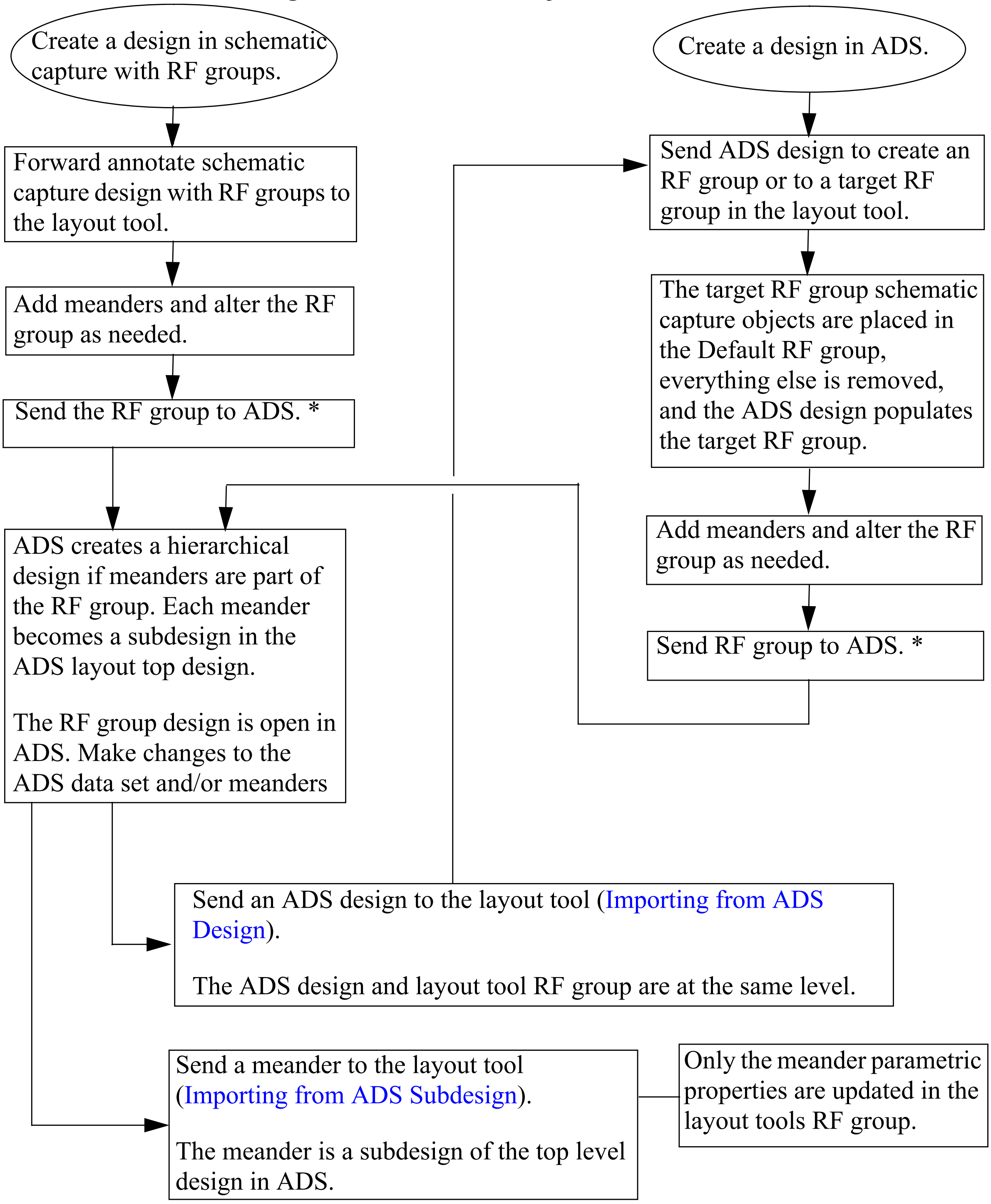
Related Topics

RF Simulator	RF Flow Introduction

Layout Tool and ADS Workflow

Data flow between ADS and the layout tool is bi-directional. When you send an ADS design to the layout tool, the RF group contains only the ADS design. When you send a non-hierarchical RF group with meanders to ADS, ADS creates a hierarchal design in which each meander is a subdesign. Figure 9-1 shows the design creation flow between ADS and the layout tool.

Figure 9-1. ADS and Layout Tool Flow



* Data not compatible between the two tools are not maintained.

When you create a design in ADS layout and import the design data into the layout tool, a new RF group (including elements) is created. After importing the ADS data set, the contents match that of the ADS design.

When you send an ADS design to layout, the RF group is frozen with an initial reference point of (0,0). On subsequent imports, the ADS design is tagged as an imported group. You can move, rotate, and connect the RF group to other objects (meanders) in the layout tool.

The ADS design is the master design for the RF group. When you send an ADS design as an update to a layout design, it replaces the contents of the RF group each time. After you send the ADS design to the layout tool, the contents match the contents of the ADS design. RF groups being updated by the ADS design maintain their RF group properties (Figure 9-1).

The following conditions apply when you change or unfreeze an RF group in the layout tool:

- ADS contains the master RF design and updates the RF group with the ADS data set. Refer to Importing from ADS Design and Importing from ADS Subdesign.
- The layout RF group becomes the "target group" (Importing from ADS Design Component Creation).

  A "target group" is an existing RF group in the layout tool being updated with the ADS data set contents.
- You can update the ADS design with your layout RF group changes using Send Design Data to ADS Simulator. You can re-send the ADS design into the layout tool.

Related Topics

ADS RF Simulator	RF Simulator
Post Back to Schematic Capture and Layout	RF Flow Introduction

Importing from ADS Design

Note
Unless otherwise stated, designs are sent from the top of the ADS hierarchy.

When you send an ADS design, the top level ADS design data is sent to the layout tool, where it creates or updates an RF group (target group). When you send a top level ADS design, the following restrictions and conditions:

- The ADS design must have a name other than “Default”.
- The ADS design component properties apply to the target group after import.
- If the target group objects cannot be moved to the Default group, the import is aborted.
- If ADS design cannot update a target RF group meander, the import is aborted.
- All objects of type “layout only” (conductive shapes, plane shapes, obstructs, vias and so on) in the target RF group are replaced by the imported design.
- All existing schematic objects (library parts and RF elements) in the target RF group are moved to the Default group and unplaced. User data such as entry rules, RF clearances, edge nodes, RFPorts, and so on, are contained in the Default group.
- All RF group meanders that have matching reference designator property values remain in the RF group. All other meanders are deleted.

Related Topics

RF Simulator
Auto Arranger
Post Back to Schematic Capture and Layout
RF Flow Introduction

Importing from ADS Subdesign

When you import from the ADS subdesign you send an ADS meander to an RF group. RF groups containing meander(s) create hierarchical designs in ADS. Each meander becomes a subdesign of the top-level ADS design. You push into an ADS subdesign to send the meander to the layout tool.

Importing an ADS meander subdesign to the layout tool only updates parametric properties and has the following ramifications:

- Nothing is deleted from the target group.
- The target group meander reference designators are updated with the ADS design parametric properties and re-synthesized.
- All user data (entry rules, RF clearances and others) are preserved.

- Only one meander can be sent at a time.

Related Topics

RF Simulator	Auto Arranger
Post Back to Schematic Capture and Layout	RF Flow Introduction

Importing from ADS Design Component Creation

When you import a top ADS design into the layout tool:

- All layout objects are created on the design.

Note
If a component cannot be created in the RF group, an error message is posted and the import continues.

- All schematic objects and elements are placed on the RF group/target.
- All existing meanders are updated with new parametric properties and re-synthesized.
- All new RFShapes are created as “layout only” elements.

Note
If the RF group/target group cannot be frozen after import, the import is aborted and no changes are made to the RF group/target group.

Related Topics

RF Simulator	Auto Arranger
Post Back to Schematic Capture and Layout	RF Flow Introduction

Installing the MentorDA_DK Design Kit

The MentorDA_DK (Mentor Dynamic Access_Design Kit) creates the ADS MentorDA menu selection (MentorDA Dropdown Selections).

Use the menu to open the ADS server, enabling a bidirectional dynamic-link between the ADS server and the schematic capture and layout tools.

Note
The **MentorDA** menu is supported only on Windows platforms with Agilent’s ADS2009 or newer software.

Use the following procedure to install the Advanced Design System (ADS) MentorDA_DK software.

Prerequisites

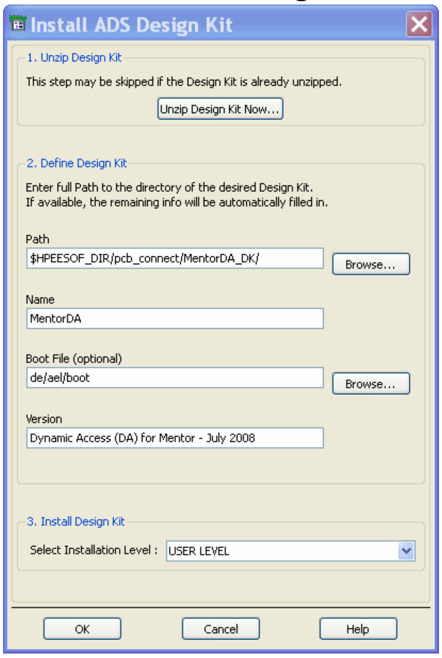
- ADS must be open.

Procedure

1. Click **DesignKit** > **Install Design Kits**.

   The Install ADS Design Kit dialog box displays (Figure 9-2).

Figure 9-2. Install ADS Design Kit Dialog Box.

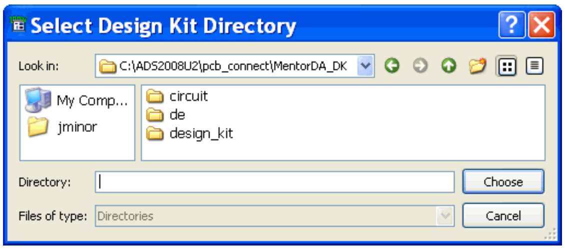


2. Select the **Browse** button for the Path field.

   Navigate to $HPEESOP_DIR\pcb_connect\MentorDA_DK, where $HPEESOP_DIR is the ADS 2009 installation directory (example C:\ADS2009\pcb_connect\MentorDA_DK).

3. Select **design_kit** (Figure 9-3).

Figure 9-3. Select Design Kit



4. Click **Choose**.
5. Select the applicable level with the **Select Installation Level** field dropdown arrow (USER LEVEL, STARTUP LEVEL, or PROJECT LEVEL)
6. Click **OK**.

Note
For additional information on the ADS RF simulator, access the ADS documentation at C:\ADS2009\doc\mentorlink.

Results

The MentorDA menu is added to the Advanced Design System (Main) dialog box.

Related Topics

Launching the ADS Server	Connecting Schematic Capture to RF Simulator Server
Connecting Layout to RF Simulator Server	ADS RF Simulator

.

MentorDA Dropdown Selections

The MentorDA dropdown menu enables you to open and close the ADS MentorDA server (Figure 9-5).

Figure 9-4. ADS/MentorDA

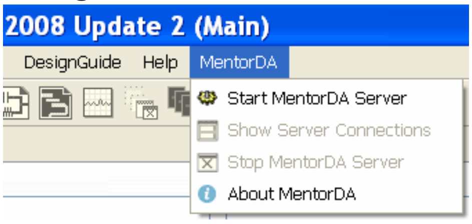


Table 9-1. MentorDA Dropdown Selections

Selection	Description
Start MentorDA Server	Opens Advanced Design System (ADS) Mentor Dynamic Access_Design Kit server.
Show Server Connections	Displays server connection information.
Stop MentorDA Server	Terminates the MentorDA server.

Usage Notes

You must open the ADS server before the schematic capture or layout tools can connect to the server (Launching the ADS Server). When the server terminates, any active connection to the schematic capture or layout tool terminate as well.

Related Topics

ADS RF Simulator

Launching the ADS Server

For interactive operation between schematic capture, layout tools, and Advanced Design System (ADS), you must open the ADS RF server.

Use the following to open the ADS server.

Prerequisites

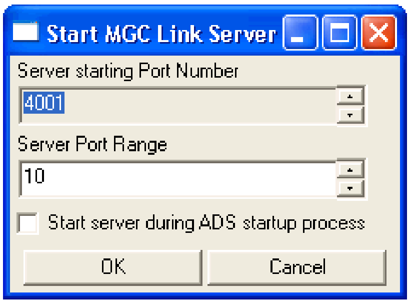
- ADS must be open.

Procedure

1. Choose **MentorDA > Start MentorDA Server**.

   The default values in Start MGC Link Server match the schematic capture and layout tools default values **(**Figure 9-5**)**.

Figure 9-5. Start MGC Link Server Dialog Box



2. Click **OK**.

   The schematic capture and layout tools can connect to any number of ADS servers. This enables you to send an RF group from the schematic capture or layout tools to any ADS workstation.

   Use the following to establish a dynamic-link:

   - Connecting Schematic Capture to RF Simulator Server
   - Connecting Layout to RF Simulator Server.

Results

The ADS Connect dialog box displays.

Related Topics

RF Group Elements Exported to RF Simulator   Layout Tool and ADS Workflow

ADS RF Simulator

Send Design Data to ADS Simulator

Sending design data requires an active dynamic-link between the Advanced Design System (ADS) RF server and the schematic capture and/or layout tools.

Use the following to transfer data between schematic capture and ADS or layout and ADS.

- Launching the ADS Server
- Connecting Schematic Capture to RF Simulator Server,
- Connecting Layout to RF Simulator Server
- Sending Netlist - Schematic Capture
- Sending Netlist - Layout

- Sending Layout Data
- Sending Tune/Opt Data

Related Topics

Generate Data ADS RF Simulator

Sending Netlist - Schematic Capture

Use the following procedure to send schematic capture netlist to the ADS simulator.

Prerequisites

- An ADS schematic must be open.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
  - Launching the ADS Server

Connecting Schematic Capture to RF Simulator Server**Procedure**

1. Click **Group/Ungroup** ( ).
2. Select an RF group.
3. Choose **Send netlist**.

Results

The schematic capture RF group netlist is active in ADS schematic.

Related Topics

Send Design Data to ADS Simulator ADS RF Simulator

Sending Netlist - Layout

Use the following procedure to send a layout RF group netlist to the ADS simulator.

Prerequisites

- An ADS schematic must be open.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
  - Launching the ADS Server

- Connecting Layout to RF Simulator Server

Procedure

1. Click **Control pane** ( ) and click the RF Design tab or **Group/Ungroup** ( ).
2. Select an RF group.
3. Choose **Send netlist**.

   The generated *.net* and *.iff* files are stored in the following locations:

```
<design_name>\RF\netlist\html\<project_name>_yyyymmdd_hhmmss.html
<design_name>\RF\netlist\xml<project_name>_yyyymmdd_hhmmss.xml
<design_name>\RF\netlist\<project_name>_yyyymmdd_hhmmss.net
```

   The generated schematic files are stored in the following locations:

```
<design_name>\RF\xml\<project_name>_yyyymmdd_hhmmss.xml
<design_name>\RF\iff\<project_name>_yyyymmdd_hhmmss.iff
```

Results

The layout RF group netlist is active in ADS layout.

Related Topics

Send Design Data to ADS Simulator   ADS RF Simulator

Sending Layout Data

Note
Simulation results may require design changes (Layout Tool and ADS Workflow).

Use the following procedure to send a layout RF group to the ADS simulator.

Prerequisites

- An ADS layout design must be open.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
  - Launching the ADS Server
  - Connecting Layout to RF Simulator Server

Procedure

1. Click **Control panel** ( ) and click the RF Design tab or **Group/Ungroup** ( ).
2. Select an RF group.
3. Choose **Send layout data**.

Results

The layout RF group data is active in ADS layout.

Related Topics

Send Design Data to ADS Simulator   ADS RF Simulator

Sending Tune/Opt Data

Note
Simulation results may require design changes (Layout Tool and ADS Workflow).

Use the following procedure to send layout RF group to the ADS simulator.

Prerequisites

- An ADS Project Open must be open.
- A meander instance name or symbol name within an RF group cannot have the same name as the RF group.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
    - Launching the ADS Server
    - Connecting Layout to RF Simulator Server

Procedure

1. Click **Control pane** ( ) and click the RF Design tab or **Group/Ungroup** ( ).
2. Select an RF group.
3. Choose **Send Tune/Opt Data**.

Note
Refer to the ADS online documentation for additional information.

In order to simulate meanders, the software must convert them to ADS models (MLIN, MCURVE, MBEND, and so on). When you send updates back to layout, these models are converted back to meanders for the layout tool.

ADS parameter updates occur manually or through the tuning/optimization routine. ADS tuning/optimization routines require you to create a schematic with the Generate/Update Schematic option.

Results

The layout models display in ADS project.

Related Topics

Send Design Data to ADS Simulator Generate Data

Generate Data

You can view the design data without enabling an active connection.

Use the following to generate design data:

- Generating Netlist
- Generating Layout Data
- Generating Tune/Opt Data

Related Topics

RF Simulator	Send Design Data to ADS Simulator

Generating Netlist

The Generating Netlist selection creates a netlist of the RF groups in multiple file formats.

Use the following procedure to generate a layout netlist file.

Prerequisites

- An ADS Project Open must be open.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
    - o Launching the ADS Server
    - o Connecting Layout to RF Simulator Server

Procedure

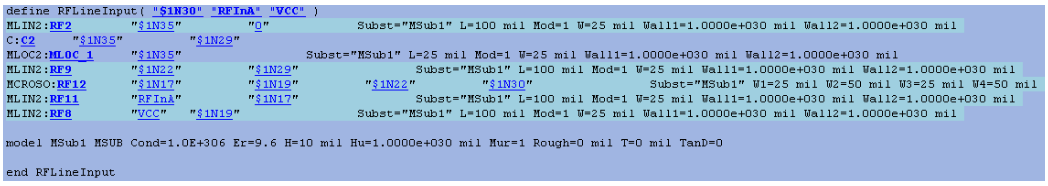
1. Click **Control pane** (), and click the RF Design tab or **Group/Ungroup** ().
2. Select the RF group.
3. Choose **Generate Netlist**.
    - Generates a compatible netlist for the RF group and subgroups.
    - Displays the netlist file in an ASCII viewer (Figure 9-6).
    - Creates the following files in the netlist folders:

<*project_directory*>\RF\netlist\html\<*design_name*>_<*group_name*>_<*yyyymmdd*>\<*hhmmss*>.html

<*project_directory*>\RF\netlist\xml\<*design_name*>_<*group_name*>_<*yyyymmdd*>\<*hhmmss*>.xml

<*project_directory*>\RF\netlist\<*design_name*>_<*group_name*>_<*yyyymmdd*>\<*hhmmss*>.net

Figure 9-6. Generate Netlist Data



```
define RFLineInput( "$1N30" "RFInA" "VCC" )
MLIN2:RF2        "$1N35"        "0"        Subst="MSub1" L=100 mil Mod=1 W=25 mil Wall1=1.0000e+030 mil Wall2=1.0000e+030 mil
C:C2     "$1N35"        "$1N29"
MLOC2:MLOC_1     "$1N35"        Subst="MSub1" L=25 mil Mod=1 W=25 mil Wall1=1.0000e+030 mil Wall2=1.0000e+030 mil
MLIN2:RF9        "$1N22"        "$1N29"        Subst="MSub1" L=100 mil Mod=1 W=25 mil Wall1=1.0000e+030 mil Wall2=1.0000e+030 mil
MCROSO:RF12      "$1N17"        "$1N19"        "$1N22"        "$1N30"        Subst="MSub1" W1=25 mil W2=50 mil W3=25 mil W4=50 mil
MLIN2:RF11       "RFInA"        "$1N17"        Subst="MSub1" L=100 mil Mod=1 W=25 mil Wall1=1.0000e+030 mil Wall2=1.0000e+030 mil
MLIN2:RF8        "VCC"   "$1N19"        Subst="MSub1" L=100 mil Mod=1 W=25 mil Wall1=1.0000e+030 mil Wall2=1.0000e+030 mil

model MSub1 MSUB Cond=1.0E+306 Er=9.6 H=10 mil Hu=1.0000e+030 mil Mur=1 Rough=0 mil T=0 mil TanD=0

end RFLineInput
```

Results

Creation of the netlist folder and files.

Related Topics

Generate Data | Send Design Data to ADS Simulator

Generating Layout Data

The Generating Layout Data selection generates iff and momentum folders.

Use the following procedure to generate layout data.

Prerequisites

- ADS Project Open must be open.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
    - Launching the ADS Server
    - Connecting Layout to RF Simulator Server

Procedure

1. Click **Control pane** ( ), and click the RF Design tab or **Group/Ungroup** ( ).
2. Select an RF group.
3. Choose **Generate Layout data**.
    - Generates iff and momentum folders for the RF group and subgroups.
    - Displays momentum _a file in the ASCII default viewer (Figure 9-7).
    - Creates the following files in the momentum and iff folders:

    Momentum files (passive circuit analysis):

```
<project_directory>\RF\momentum\html\<design_name>_<group_name>_<yyyymmdd
>\<hhmmss>.html

<project_directory>\RF\momentum\xml\<design_name>_<group_name>_<yyyymmdd>
\<hhmmss>.xml

<project_directory>\RF\momentum\<design_name>_<group_name>_<yyyymmdd>\<hh
mmss>_a.
```

Figure 9-7. Generate Layout Data Text File

```
UNITS MIL,10000;
GRID 5.00,2 0.00,0.00;
LOCK 0.00;
TSLANT 0.00;
RESOLUTION 0.00;
$$
$$ layer line
$$ label type type color pen layer
$$
EQU Microstrip_1        :D :L1    :C52      :P1  52     ;
EQU Plane_2             :D :L1    :C53      :P1  53     ;
EQU Plane_3             :D :L1    :C54      :P1  54     ;
EQU Stripline_4         :D :L1    :C55      :P1  55     ;
EQU Stripline_5         :D :L1    :C56      :P1  56     ;
EQU Stripline_6         :D :L1    :C57      :P1  57     ;
EQU Stripline_7         :D :L1    :C58      :P1  58     ;
EQU Microstrip_8        :D :L1    :C59      :P1  59     ;
EXIT;
```

IFF file (export design data):

<*project_directory*>\RF\iff\<*design_name*>_<*group_name*>_<*yyyymmdd*>\<*hhmmss*>.iff

Results

Creation of the iff and momentum folders and files.

Related Topics

Generate Data

Send Design Data to ADS Simulator

Generating Tune/Opt Data

Use the following to generate the tune/opt data.

Prerequisites

- An ADS Project Open must be open.
- A dynamic-link must be established between the ADS server and the schematic capture tool.
  - Launching the ADS Server
  - Connecting Layout to RF Simulator Server

Procedure

1. Click **Control pane** ( ), and click the RF Design tab or **Group/Ungroup** ( ).
2. Select an RF group.
3. Choose **Generate Tune/Opt Data**.
   - Generates *.iff* file for the RF group and subgroups.
   - Creates the following files in the iff folder:

<project_directory>\RF\iff*<design_name>*_*<group_name>*_*<yyyymmdd>*_*<hhmmss>*.iff

Results

Creation of the iff folder and *.iff* file.

Related Topics

Generate Data

Send Design Data to ADS Simulator

Adding ADS RF Shapes to Central Library

The ADS RF shape library at %SDD_HOME%\standard RF\ShapesLibrary is a subset of the ADS RF library. The ShapesLibrary contains symbols that are identical to the ADS symbols and pertain to RF/Microwave designs.

You must merge the ADS Shapes Library into your Central Library (CL). Merge the libraries with a <*file_name*>.bat file or command line.

Use the following procedure to merge the ADS library with your design library.

Procedure

1. Type the following in a command prompt window:

```
%SDD_HOME\common\win32\bin>RFSymMergeUtility
```

   Command and options display:

```
RFSymMergeUtility [-ads | awr] [-rfcl  <path to RF CL/Source CL>]
[-all | (list of partition names) ]  [-update | overwrite]   -userCL
<customer-CL/Destination CL>
```

2. Enter command and options on a command line or in a <*file_name*>.bat file.

Example of a *.bat* file:

Note
The command must be on one line ,or must use the ^ continuation character.

```
C:\MentorGraphics\<EE_software>\SDD_HOME\common\win32\bin\RFSymMergeUtili
ty.bat -ads -rfcl ^
C:\MentorGraphics\<EE_software>\SDD_HOME\standard\RF\ShapesLibrary\Shapes
Library.lmc -all -update -userCL ^
U:\Design_Central_Library\Central_Library.lmc
```

1. Save <*file_name*>*.bat* file.
2. Double-click <*file_name*>*.bat* file.

Results

The command prompt window displays information about the merge as shown below.

```
C:\MentorGraphics\7.9.2EE\SDD_HOME\common\win32\bin>C:\MentorGraphics\7.9
.2EE\SDD_HOME\common\win32\bin\RFSymMergeUtility.bat -ads -rfcl
C:\MentorGraphics\7.9.2EE\SDD_HOME\standard\RF\ShapesLibrary\ShapesLibrar
y.lmc -all -update -userCL U:\Design_Central_Library\Central_Library.lmc
Source
CL:C:\MentorGraphics\7.9.2EE\SDD_HOME\standard\RF\ShapesLibrary\ShapesL
ibrary.lmc
```

```
 Destination CL:U:\Design_Central_Library\Central_Library.lmc
 Patitions to update:
-all
mode: append
Copying partition RF_Data_Items
Copying partition RF_Devices_BJT
Copying partition RF_Devices_GaAs
Copying partition RF_Devices_JFET
Copying partition RF_Devices_Linear
Copying partition RF_Devices_MOS
Copying partition RF_Eqn_Based_Nonlinear
Copying partition RF_Filters_Lowpass
Copying partition RF_Linear_Data_File
Copying partition RF_Lumped_Components
Copying partition RF_Lumped_With_Artwork
Copying partition RF_Passive_RF_Circuit
Copying partition RF_Simulation_AC
Copying partition RF_Simulation_DC
Copying partition RF_Sources_Time_Domain
Copying partition RF_System_PLL_components
Copying partition RF_TLines_Finline
Copying partition RF_TLines_Ideal
Copying partition RF_TLines_Microstrip
Copying partition RF_TLines_Multilayer
Copying partition RF_TLines_Printed_Circuit_Board
Copying partition RF_TLines_Stripline
Copying partition RF_TLines_Suspended_Substrate
Copying partition RF_TLines_Waveguide
Copying partition RF_UTILS
Copying partition RF_Varblocks

C:\MentorGraphics\7.9.2EE\SDD_HOME\common\win32\bin>
```

Related Topics

Sending ADS Design to Schematic Capture Tool

Send Design Data to ADS Simulator

Connecting Schematic Capture to RF Simulator Server

Sending ADS Design to Schematic Capture Tool

Designs are sent to ADS from the schematic capture tool over the dynamic-link. You can modify ADS RF groups and update the schematic capture design.

Caution

When you create your design in ADS, you must use a net to connect the parts. Part-to-part connections are not supported in the schematic capture tool (RF Simulator Design Generation).

Prerequisites

- A schematic must be open in the schematic capture tool.
- You must connect the schematic capture tool to the RF simulator server (Connecting Schematic Capture to RF Simulator Server).
- Each ADS port must have a unique number, starting at 1 and incrementing.

Use the following to send ADS data to the schematic capture tool.

Procedure

1. Choose **MentorDA > Send ADS Schematic**.

   The ADS schematic attaches to the schematic capture tool cursor.

2. Click to place the schematic.

Results

The ADS schematic appears in the schematic capture tool.

Related Topics

ADS RF Simulator

Launching the ADS Server

Send Design Data to ADS Simulator

Sending ADS RF Design to Layout Tool

Use the following procedure to send an ADS design to the layout tool.

Prerequisites

- ADS must have a layout design open (refer to ADS documentation).
- The layout tool must have an open design.
- You must connect the layout tool to the RF simulator (Connecting Layout to RF Simulator Server).

Procedure

1. Select the RF group in ADS Layout.
2. Choose **MentorDA** > **Send ADS Layout**.

   The RF Mapping rflayermap.xml Dialog Box opens in the layout tool.
3. Select RF Mapping rflayermap.xml dialog box fields as required (Creating an ADS Mapping File).
4. Click **OK**. The software does the following:
   - Freezes the RF group in the layout tool (Freeze/Unfreeze an RF Group).
   - Assigns the GROUP_IMPORTED property to the RF group (RF Control Dialog Box, RF group tab).
   - Unfreezes the RF group so you can make changes.

Note

If you import the ADS group again, a messages appears warning you that your RF group layout tool changes will be lost if you continue.

   - Updates a frozen GROUP_IMPORTED RF group in the layout tool, by making modification in ADS and resending the RF group to the layout tool. When you send the ADS RF group to the layout tool, the software unfreezes, updates the RF group with the ADS changes, and refreezes the RF group.

Results

The RF group in layout reflects the ADS layout RF design.

Related Topics

Send Design Data to ADS Simulator

Sending ADS RF Design to Layout Tool

Using Extended Configuration, Enable Cutting and Default Clearance

Layout Tool and ADS Workflow

RF Mapping rflayermap.xml Dialog Box

To access:

- Click **Set mapping** button in the RF Connect - Configurations tab (RF Connect Dialog Box (Layout) - Configuration Tab).
- Send ADS data to the layout tool. By default, the **Show mapping dialog during input** option is checked in the RF Connect Dialog Box (Layout) - Configuration Tab.

Map the ADS layers, vias, and pins to the layout design with the RF Mapping rflayermap.xml dialog box (Figure 9-8).

Caution

In ADS, any shape that is on a via layer and mapped to a padstack in the RF Mapping rflayermap.xml dialog box, becomes a via in the layout tool (Creating an ADS Mapping File).

Figure 9-8. RF Mapping rflayermap.xml Dialog Box

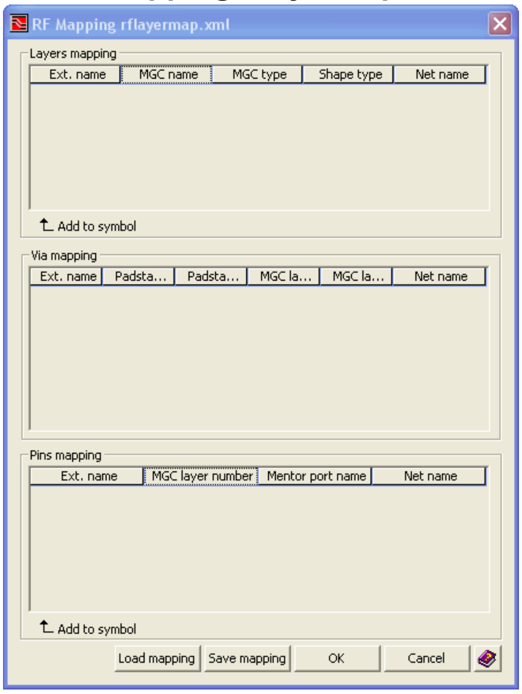


Table 9-2. RF Mapping rflayermap.xml Dialog Box Contents

Field	Description
Layers section — Maps ADS layer names to the layout tool layer names	
Ext. name (External name)	Defines the RF simulator layer name. When you send an ADS design with the dynamic-link, the RF simulator layer names automatically populate this column.
MGC name	Defines the layout tool layer name. Use the dropdown list to select the layout layer number and description. All physical and user layers are available in the list, with the following options: **No Mapping** — The data on the external layer is ignored. **Automatic** — Generates a user layer with the same name as the external layer, and all data is imported as user data.
MGC Type	Defines the layer type as Signal or Plane.

Table 9-2. RF Mapping rflayermap.xml Dialog Box Contents (cont.)

Field	Description
Shape type	Defines the type of shape. **Plane** — Polygons on the external layer (**Ext. name**) are imported as plane shapes on the MGC layer and assigned to the net in the net column. **Conductive** — Polygons on the external layer are imported as conductive shapes. You can assign a net in the net name column. **Placement Obstruct Top/Bottom** — Polygons on the external layer are imported as placement obstructs on the top or bottom layer. **Route Obstruct** — Polygons on the external layer are imported as route obstructs. **RF Shape** —Polygons on the external layer are imported as a custom RF Shape. **Plane Obstruct**: Polygons on the external layer are imported as plane obstructs. **Pad Obstruct** — Polygons on the external layer are imported as Pad obstructs
Net name	Defines the net associated with the MGC Type.
Via mapping section — Maps ADS via names to the layout tool via names.	
Ext. name (External name)	Defines the RF simulator via name.
Padstack name	Defines the layout tools padstack name. Use the dropdown list to select a padstack.
Padstack span	Defines the layer span of the padstack in the layout tool.
MGC layer start	Defines the start layer for a via in the layout tool
MGC layer end	Defines the end layer for a via in the layout tool.
Net name	Defines the net name assigned to the via. Use the dropdown list to select a net name in the layout tool.
Pins mapping section — Maps ADS pin names to the layout tool pin names	
Ext. name (External name)	Defines the name of the port or pin in the RF simulator design.
MGC layer number	Defines the pin layer in the layout tool. This can be any physical layer in the current design. Use the dropdown list to select the layer number.
MGC port name	Defines the port name for the pin in the layout tool. Use the dropdown list to select the layout tool port name.

Table 9-2. RF Mapping rflayermap.xml Dialog Box Contents (cont.)

Field	Description
Net name	Defines the net name in the layout tool. Use the dropdown list to select a net in the layout tool.
Add to Symbol	Checked, enables column data to be included in schematic capture symbol generation, which enables placement of custom RF shapes in a schematic. This option is enabled when you assign a layout layer or pin (port).
Load mapping	Loads an existing RF Mapping file. Rows not mapped are yellow. Creates an rflayermap.xml file in the <*design_name*>\RF directory.
Save mapping	Saves your RF Mapping file in the <*design_name*>\RF directory.

Related Topics

ADS RF Simulator	Creating an ADS Mapping File
Post Back to Schematic Capture and Layout	Auto Arranger

Creating an ADS Mapping File

Create a mapping file with the RF Mapping rflayermap.xml Dialog Box. You can define the ADS-to-layout tool relationship for layers, vias, and pins with the RF Mapping rflayermap.xml dialog box.

Use the following procedure to create an RF Mapping rflayermap when you send ADS data to the layout tool.

Prerequisites

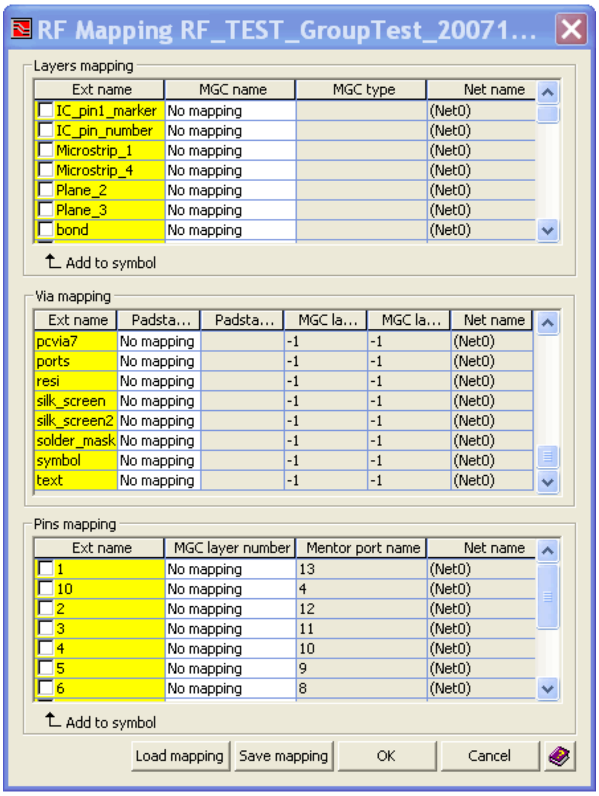
- ADS must have a layout design open (refer to ADS documentation).
- The layout tool must have a design open (RF Toolkit Toolbar).
- A dynamic-link must be established between the ADS server and the schematic capture tool (Connecting Layout to RF Simulator Server).
- “Show mapping dialog during input” must be checked in the RF Connect Dialog Box (Layout) - Configuration Tab.

Procedure

1. Select **MentorDA > Send ADS Layout** in the RF simulator Layout main menu.

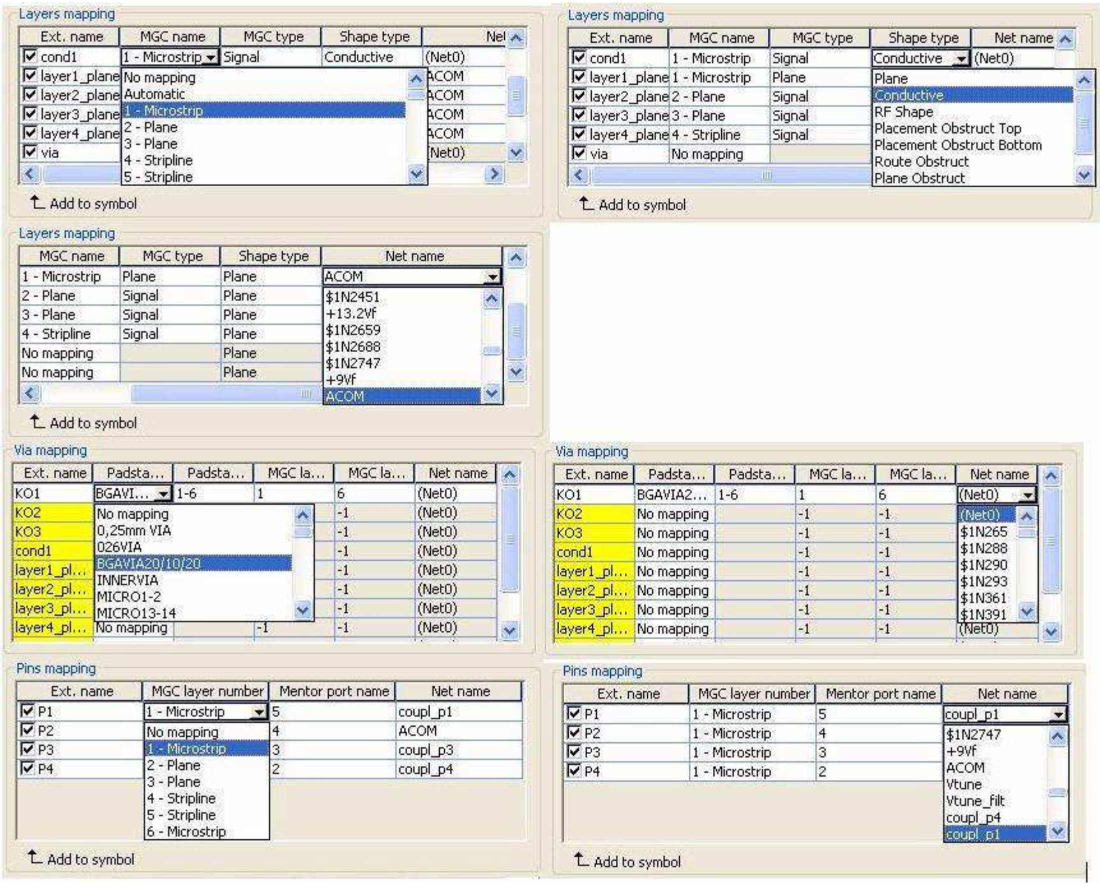
   When you send the design to layout, the RF Mapping rflayermap.xml dialog box automatically appears (Figure 9-9). If no mapping file is found, the Ext.name (External name) column appears yellow.

Figure 9-9. RF Mapping First Time



2. Use the dropdown list to assign mapping for layers, vias, and pins, as shown in Figure 9-10.

Figure 9-10. RF Mapping to Layout



3. Click **Save mapping**.

   The software creates a <*design_name*>\Pcb\<*map_file_name*>.xml file.

4. Place the ADS design in the layout tool.

Results

The ADS design appears in the layout tool.

Related Topics

Post Back to Schematic Capture and Layout

Layout Tool and ADS Workflow

Adding ADS RF Shapes to Central Library

Using Extended Configuration, Enable Cutting and Default Clearance

Use the Enable Cutting selection on the RF Connect Dialog Box (Layout) - Extended Configuration Tab to define an area within an RF group to be sent to the RF simulator.

The RF_SIMULATION_OUTLINE polygon defines the region sent to the RF simulator. When you select the Enable Cutting option, the objects within the polygon are sent to the RF simulator (Cutting).

The Default Clearance value defines the clearance area of the objects within the polygon. Use the RF_SIMULATION_OUTLINE to increase the clearance area (Clearance).

Note

The RF_SIMULATION_OUTLINE draw object must be part of the RF group.

Related Topics

Sending Layout Data

RF Simulator

Layout Tool and ADS Workflow

MentorDA Dropdown Selections

RF Flow Introduction

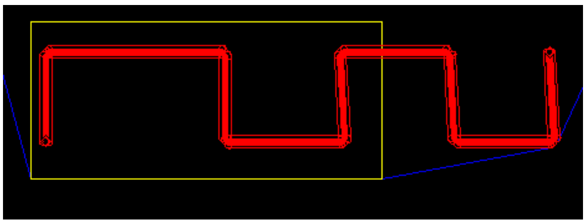
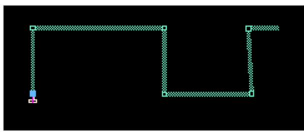
RF Connect Dialog Box (Layout) - Extended Configuration Tab

Cutting

The RF_SIMULATION_OUTLINE polygon defines the objects within the RF group to send to the RF simulator when you select **Enable Cutting** (RF Connect Dialog Box (Layout) - Extended Configuration Tab).

Figure 9-11 shows an RF_SIMULATION_OUTLINE draw object polygon (yellow) encompassing part of a meander (red). When you use the **Send layout data** command for the RF group, the RF simulator receives only the area within the polygon.

Figure 9-11. Specific Area Sent to RF Simulator

Layout selection sent	**RF simulator receives**
	

Related Topics

Using Extended Configuration, Enable Cutting and Default Clearance

Clearance

Post Back to Schematic Capture and Layout

MentorDA Dropdown Selections

Clearance

The RF_SIMULATION_OUTLINE polygon for a meander is shown in Figure 9-12 (Cutting).

The meander has two vias, 1:2 and 2:3 in the layout tool (layer 1 blue, layer 2 pink, and layer 3 green).

Figure 9-12. Layout Meander on Planes 1, 2, and 3

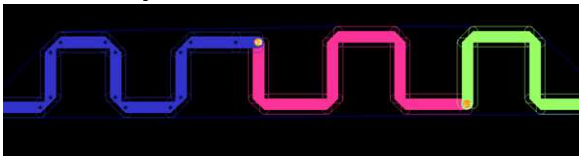


Figure 9-13 displays the layout region sent to the RF simulator with Enable Cutting defaults.

Figure 9-13. RF Simulator Display of Meander

Using Figure 9-12 as a reference, the following examples demonstrates the Enable Cutting option:

- Default Clearance and Layer Example
- Layer Clearance Example
- Layer Clearance and Over/Under Example

Default Clearance and Layer Example

In this example, the layout meander shown in Figure 9-12 has the following Enable Cutting selections as shown Figure 9-14.

Clearance — 50

Selections — Layer - 1, 2, and 3 checked

Types — Components

Figure 9-14. Meander with Default Clearance

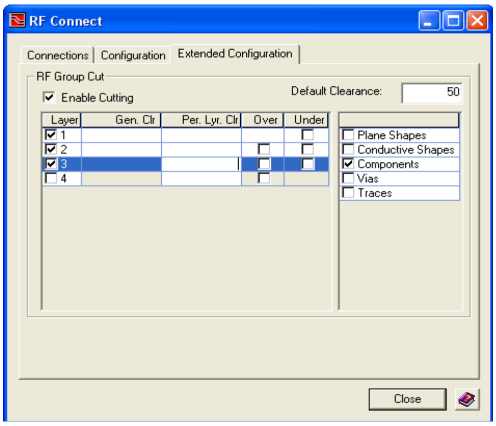


Figure 9-15 displays the ADS results after you send the layout data to ADS simulator.

Figure 9-15. RF Simulator Results with Default Clearance

Figure 9-16 shows an ADS 3D representation of the meander. The default clearance value specifies the meanders clearance display within the polygon region. The meander and default clearances for the selected layers display because of the Layer sections.

Figure 9-16. 3D Representation of Layer and Default Clearance

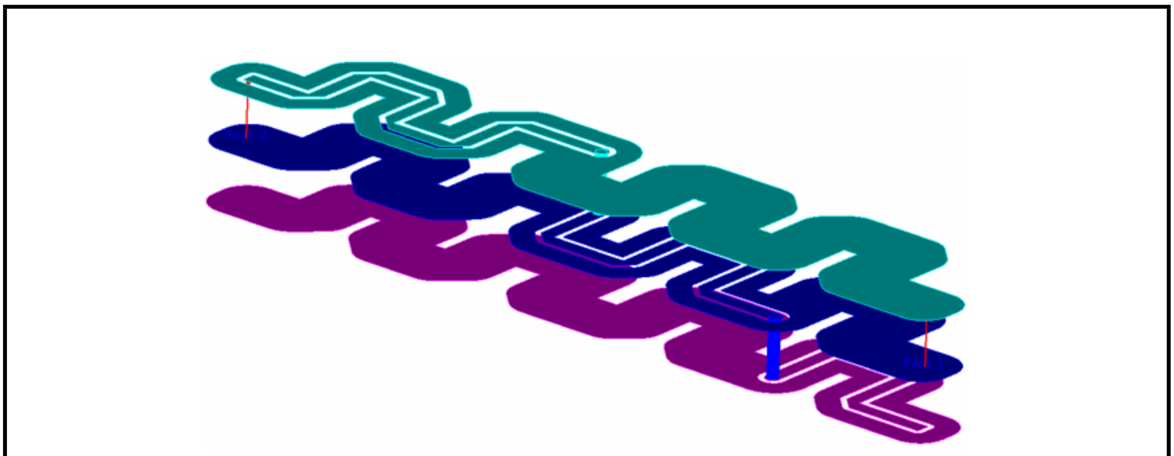


Figure provided by Advanced Design System (copyright 2008 Agilent Technologies

Layer Clearance Example

In this example, Figure 9-12 has the following Enable Cutting selections as shown Figure 9-18:

Clearance — 50

Selections — Per-layer clearance, layer 1 - 50, layer 2 - 75, layer 3 - 50

Types — Components

Figure 9-17. Trace with Clearance

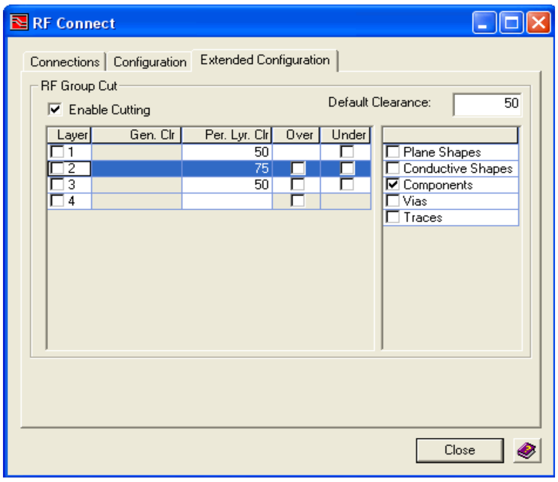
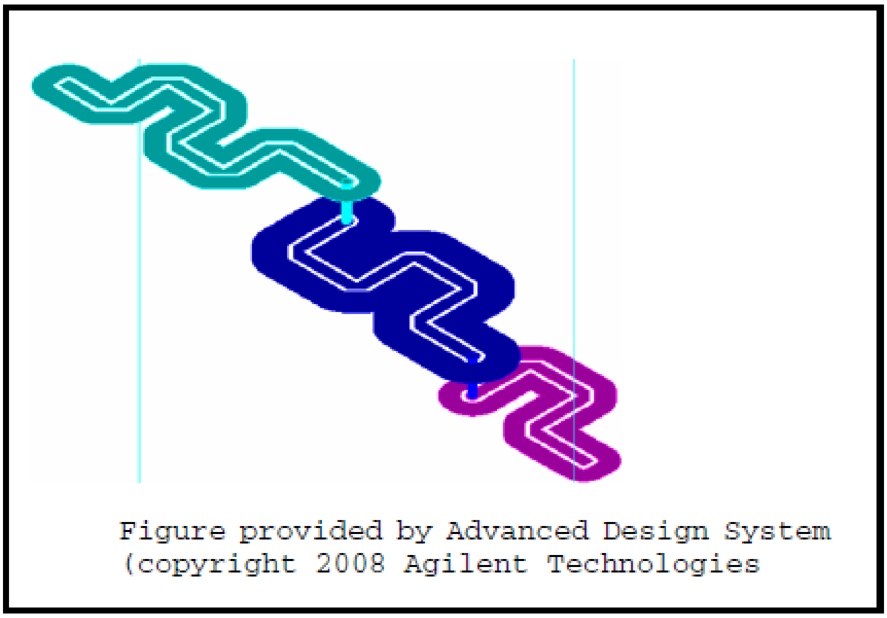


Figure 9-18 shows the results after you send layout data to the ADS simulator.

Figure 9-18. RF Simulator Results with Clearance

Figure 9-19 shows an ADS 3D representation of the meander. No Layers are checked, therefore only the meander's per-layer clearance values display on each layer.

Figure 9-19. 3D Representation for Clearance



Layer Clearance and Over/Under Example

In this example, Figure 9-12 has the following Enable Cutting selections as shown Figure 9-20:

Clearance — 50

Selections — Per-layer clearance, layer 1 - 50, layer 2 - 75, layer 3 - 50

Selections — Layer 2 with Over and Under

Types — Components

Figure 9-20. Trace with Over and Under

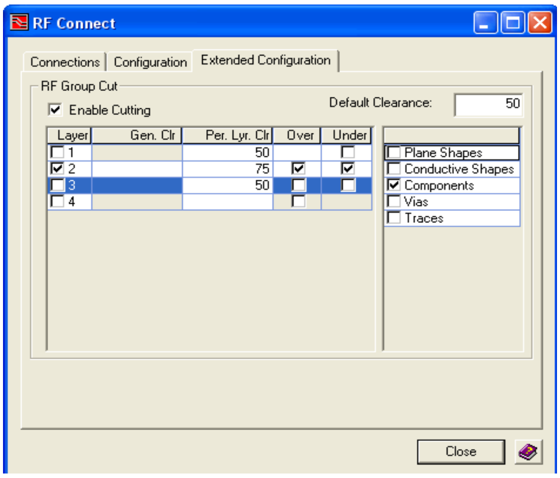


Figure 9-21 shows a 3D representation in the RF simulator. Layer 2 becomes the reference layer displaying the meander. The Over and Under sections enables you to view the area over and under the layer 2 meander with the per layer clearance.

Figure 9-21. 3D Representation for Trace Over and Under

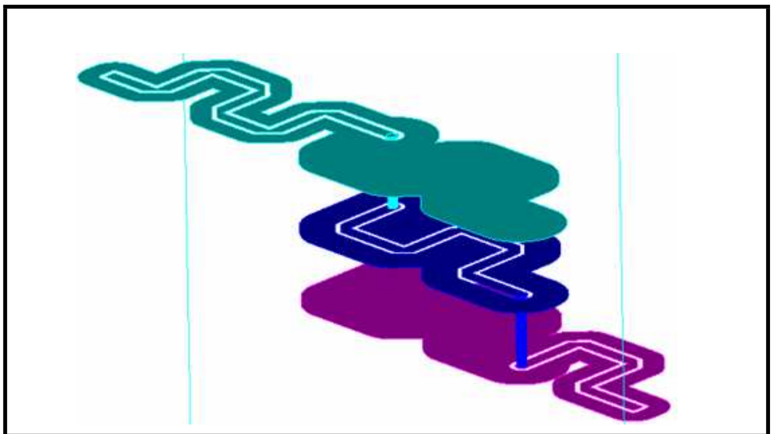


Figure provided by Advanced Design System
(copyright 2008 Agilent Technologies

Related Topics

Using Extended Configuration, Enable Cutting and Default Clearance

Clearance

Post Back to Schematic Capture and Layout

Post back enables you to send designs in the RF simulator to the schematic capture or layout tool. Use the following procedures.

- Posting Back to Schematic Capture
- Posting Back to Layout

Note

When you send a top level schematic back to schematic capture, you can only update the parameters.

After posting back a design, the Central library SymbolLibs contains updated RF_* files. which Mentor Graphics tools use.

Related Topics

Launching the ADS Server

MentorDA Dropdown Selections

Auto Arranger

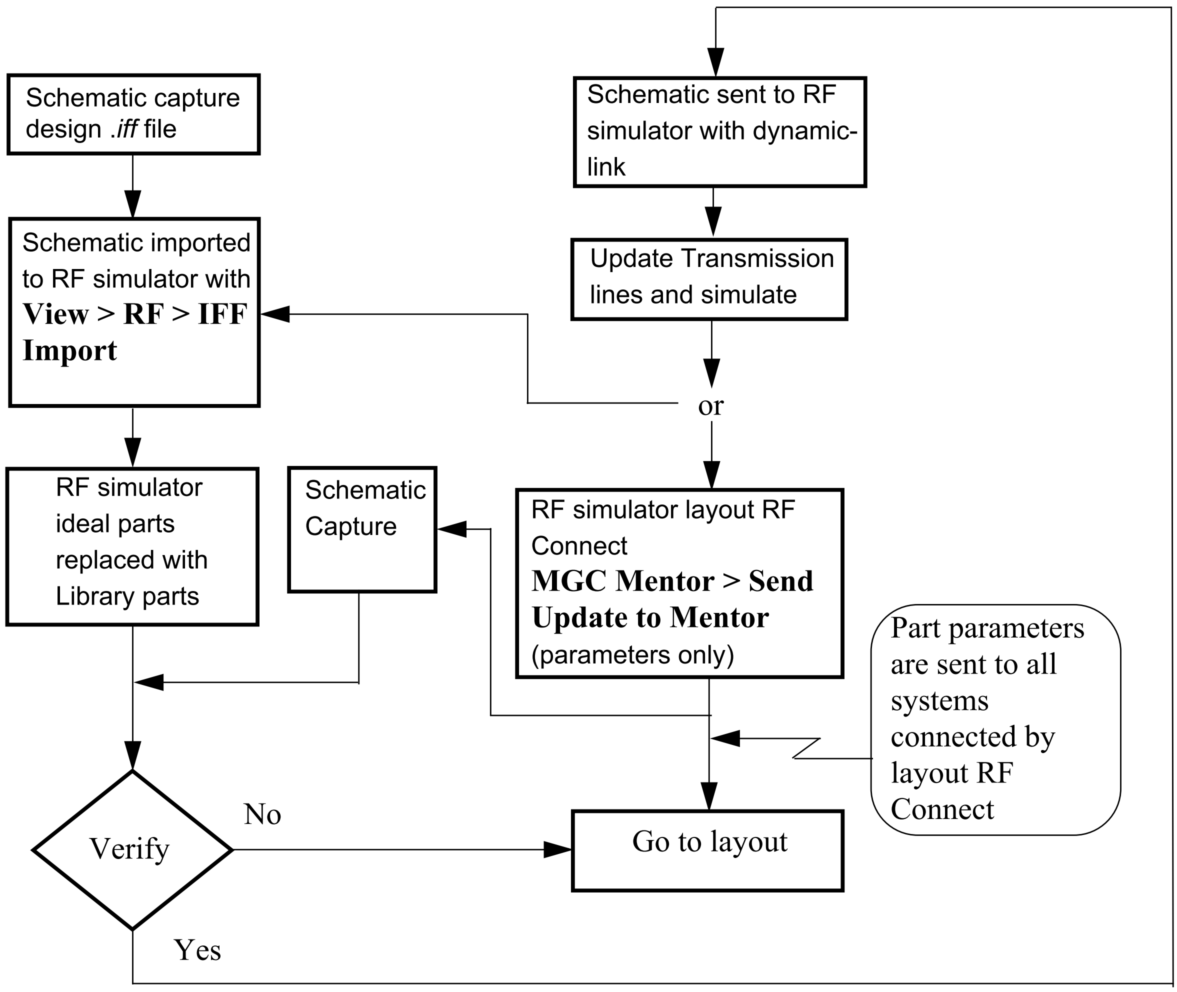
User-defined RF Shapes

Posting Back to Schematic Capture

Post Back enables you to send an RF simulator schematic design to a Mentor Graphics schematic capture tool. The Post Back process flow is shown in Figure 9-22. In order to Post Back a design, it must:

- Be created in RF simulator schematic with symbols in the Central Library (Adding ADS RF Shapes to Central Library).
- Be Flat or Flattened to avoid hierarchy.
- Be a design that was sent from the schematic capture tool to the RF simulator, altered in RF simulator, and sent back to the schematic capture tool.

Figure 9-22. Circuit Created in RF Simulator Flow



To post back a design from the RF simulator layout tool to the schematic capture tool, use the following procedure.

Prerequisites

- The design must be open in the schematic capture tool.
- The design must be open in ADS.

Procedure

Note

If you are using ADS, all shapes must be filled. To fill a shape, select the shape in ADS, then right-click **Modify** > **Join**.

1. Choose **File** > **Export** and create an *.iff* (Interchange File Format) file in the RF simulator schematic window.
2. Choose **View** > **RF** > **IFF Schematic** in the Mentor Graphics schematic capture tool.
3. Select the *.iff* file generated by the RF simulator in the Open dialog box.

   If the schematic capture tool already contains a design with the same name as the *.iff* file (for example *balmix* in Figure 9-23), select one of the following from the IFF Import dialog box:

   a. **Update parameters** — Only updates the RF parameters on existing shapes in the current schematic.

   b. **Replace** — Replaces the existing schematic with the schematic from the *.iff* file.

Figure 9-23. IFF Import Dialog Box

4. Import the *.iff* file into schematic capture. You **must** replace the symbols with schematic capture Central Library parts.
5. Replace parts and send the schematic back to the RF simulator for re-evaluation with RF connect.

Note

Only capacitors, resistors, transistors, and diodes are supported with the RF Connect link.

In the schematic capture tool, you can also generate an *.iff* file by selecting Generate Schematic Data. The location of the generated *.iff* file is:

```
<project directory>\RF\iff\<project name>_<current_date>_<current
time>.iff
```

You can also generate an *.iff* file that can be imported into the RF simulator. In the schematic capture tool, choose **Generate Schematic Data**. File location is:

6. Choose **File > Import** and open the schematic capture *.iff* file in the RF simulator.
7. Modify the transmission line RF symbols to those used in the RF simulator.
8. Post Back parameter changes for capacitors, resistors, transistors, and diodes, by performing a *.iff* file transfer, or choose **MentorDA > Send Update to Mentor** in the RF simulator.

   The RF simulator can update the parameters only if the schematic capture and layout tools are connected through the RF Connect link.

   The RF simulator and schematic capture and layout flow is shown in Figure 9-22.

Results

The RF simulator design appears in the schematic capture tool.

Related Topics

Post Back to Schematic Capture and Layout

Posting Back to Layout

Sending ADS Design to Schematic Capture Tool

Posting Back to Layout

Post Back enables you to send an RF simulator layout design to the Mentor Graphics layout tool where you can finalize and save it as a custom RF shape. If you do not save the design as a custom RF shape, it only exists in the current layout.

The RF simulator **MentorDA > Send Layout Update to Mentor** selection enables you to use the custom shape in another design by generating a symbol for it, instantiating it in a schematic, and forward annotating.

The Post Back layers, vias, and ports are controlled by a mapping file (Creating an ADS Mapping File).

Use the following procedure to Post Back a design in the RF simulator to the layout tool.

Procedure

1. Create an RF connection between the RF simulator and the Mentor Graphics layout tool (Connecting Layout to RF Simulator Server).
2. Select the design to be posted back in the RF simulator layout window.
3. Choose **MentorDA > Send Layout Update to Mentor**.

   You can use a mapping file to control the mapping of the RF simulator layers and vias to the Mentor Graphics layout tool (RF Mapping rflayermap.xml Dialog Box).

   The RF simulator design is imported into an RF group, using the name assigned to the the RF simulator project. The group is initially located at the origin, and you can move it as with any other group. If the procedure is repeated after the group has been moved, the last location of the shape is preserved.
4. Create the library symbol (Creating a Library Part).

   The shape definition is stored in the library elements file, and the symbol is stored in the $MGC_ADSLIB directory.

Results

The RF Control dialog box, RF Library Shape tab - User directory, lists the new RF shape.

Related Topics

Send Design Data to ADS Simulator	Sending ADS RF Design to Layout Tool
Adding ADS RF Shapes to Central Library	Posting Back to Schematic Capture

Chapter 10
MWO RF Simulator

MWO RF Simulator Introduction

The schematic capture and layout tools use the RF simulator named Applied Wave Research (AWR) Microwave Office (MWO) to simulate RF groups.

Caution

Install all Mentor Graphics software and relevant patches before you install AWR/MWO and AWRConnect MentorSetup software. If you install Mentor Graphics tools after installing AWRConnect MentorSetup software, remove the AWRConnect MentorSetup software and reinstall (Installing MWO Connected MentorSetup).

Use the following procedures as needed:

- Installing MWO Connected MentorSetup
- Adding AWR/MWO RF Shapes to Central Library
- Synchronizing AWR/MWO and Central Library Libraries
- Launching MWO Server
- Sending Schematic Capture RF Shapes
- Sending an MWO Design to Schematic Capture
- Sending Layout Data
- Posting Back MWO Updates to Layout

Related Topics

RF Simulator	RF Layout
Getting Started	Schematic Capture

Installing MWO Connected MentorSetup

The AWRConnected MentorSetup wizard creates a menu selection, enabling you to launch the AWR/MWO RF simulator server (Launching MWO Server). The schematic capture and layout tools use RF Connect to connect to the AWR/MWO server creating the dynamic-link.

(Connecting Schematic Capture to RF Simulator Server and Connecting Layout to RF Simulator Server).

Note

When you install AWR, schematic capture, and layout on different workstations, the schematic capture and layout tools workstation must have the environment variable AWRD_LICENSE _FILE set to the remote AWR license file. For more information about the environment variable, refer to the AWR documentation.

Use the following procedure to install AWRConnected MentorSetup software.

Prerequisites

Caution

If you update the Mentor Graphic software after installing the AWRConnected MentorSetup wizard, uninstall and then reinstall the wizard.

- Mentor Graphics schematic capture and layout tool must be installed.
- AWR/MWO software must be installed.

Procedure

Note

The AWRConnected MentorSetup wizard is only supported on Windows platforms.

1. Unzip the *AWRConnectedMentorSteup_xxx*.zip file.
2. Open the *AWRConnected MentorSetup*.msi file.
3. Click **Next**.
4. Define **Folder** and set to **Self** or **Everyone**.
5. Click **Next**.
6. Click **Next** to confirm installation.
7. Click **Close** in the Installation Complete window.
8. Confirm that the menu is available, choose **View > Project Browser > Wizard > AWRConnected Mentor** menu selections.

Related Topics

Adding AWR/MWO RF Shapes to Central Library

Launching MWO Server

RF Layout RF Simulator

Adding AWR/MWO RF Shapes to Central Library

The AWRShapesLibrary contains a subset of the AWR/MWO RF shape library. The AWRShapesLibrary contains symbols that are identical to AWR/MWO symbols, which pertain to RF/Microwave designs.

You must merge the AWRShapesLibrary into your Central library (CL) to place RF symbols in the schematic capture tool (Placing RF Symbols).

Use the following procedure to merge the AWR/MWO library into your Central library.

Procedure

1. Open a Command Prompt window.
2. Type the following command in a Command Prompt window:

```
%SDD_HOME\common\win32\bin>RFSymMergeUtility
```

   The following command and options display:

```
RFSymMergeUtility [-ads | awr] [-rfcl  <path to RF CL/Source CL>]
[-all | (list of partition names) ]  [-update | overwrite]   -userCL
<customer-CL/Destination CL>
```

3. Enter the command and options on a command line or in a <*file_name*>.bat file.

   Example:

Note
The command must be on one line or use the ^ as a continuation character.

```
C:\MentorGraphics\<EE_software>\SDD_HOME\common\win32\bin\RFSymMergeUtili
ty.bat -awr -rfcl ^
C:\MentorGraphics\<EE_software>\SDD_HOME\standard\RF\AWRShapesLibrary\AWR
ShapesLibrary.lmc -all -update -userCL ^
U:\Design_Central_Library\Central_Library.lmc
```

4. Save <*file_name*>*.bat* file.
5. Double-click <*file_name*>*.ba*t file.

   The Command Prompt window displays information about the merge.

```
C:\MentorGraphics\2007.6EE\SDD_HOME\common\win32\bin\RFSymMergeUtility.ba
t -awr -rfcl
C:\MentorGraphics\2007.6EE\SDD_HOME\standard\RF\AWRShapesLibrary\AWRShape
sLibrary.lmc RF* -update -userCL
U:\Central_Library_Vidar_RF\Central_Library.lmc
```

```
Source
CL:C:\MentorGraphics\2007.6EE\SDD_HOME\standard\RF\AWRShapesLibrary\AWRS
hapesLibrary.lmc
Destination CL:U:\Central_Library_Vidar_RF\Central_Library.lmc
Patitions to update:
RF*
mode: append
Copying partition RF_MWO_Microstrip_Bends
Copying partition RF_MWO_Microstrip_Components
Copying partition RF_MWO_Microstrip_Coupled_Lines
Copying partition RF_MWO_Microstrip_Junctions
Copying partition RF_MWO_Microstrip_Lines
Copying partition RF_MWO_Microstrip_Other
Copying partition RF_MWO_Simulation_Control
Copying partition RF_MWO_Stripline_Bends
Copying partition RF_MWO_Stripline_Components
Copying partition RF_MWO_Stripline_Coupled_Lines
Copying partition RF_MWO_Stripline_Junctions
Copying partition RF_MWO_Stripline_Lines
Copying partition RF_MWO_Stripline_Other
Copying partition RF_MWO_Substrates
Copying partition RF_UTILS
Copying partition RF_Varblocks
C:\Documents and Settings\jminor>
```

Results

When the command completes, the RF simulator and Central library are synchronized.

Related Topics

Launching MWO Server	Send Design Data to MWO Simulator
RF Simulator	RF Layout
Getting Started	Schematic Capture

Synchronizing AWR/MWO and Central Library Libraries

After you add RF shapes to the MWO Elements Browser selection, you must add the shapes to the Central library.

Use the schematic capture tool to update the Central library with the new AWR RF shapes.

Prerequisites

- The schematic capture tool and AWR must have an active dynamic-link (Launching MWO Server and Sending Schematic Capture RF Shapes).

Procedure

1. Open the RF Group/Ungroup Dialog Box in schematic capture tool.
2. Select an RF group.
3. Choose **Import Library**.

Results

When **Import Library** completes, the RF simulator and Central library are synchronized.

Related Topics

Launching MWO Server	Send Design Data to MWO Simulator
RF Simulator	RF Layout
Getting Started	Schematic Capture

Launching MWO Server

When you launch the AWR/MWO server, you enable the schematic capture and layout tools to connect to the RF simulator server.

Use the following procedure to launch the AWR/MWO server.

Prerequisites

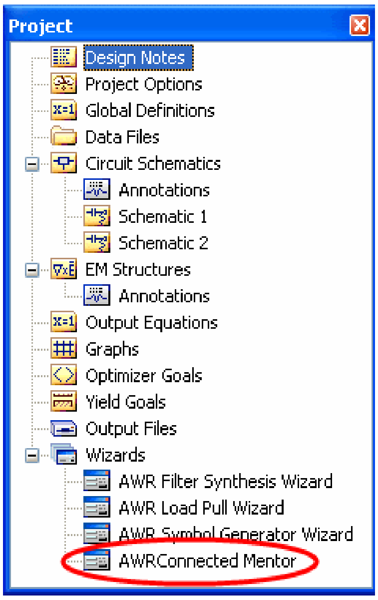
- AWR software must be installed (Installing MWO Connected MentorSetup).
- AWR must have an open schematic.

Procedure

1. Select **View > Project Browser** in the MWO window.

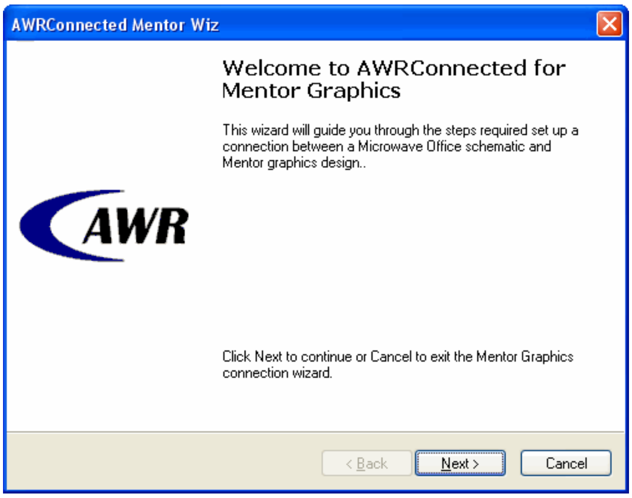
2. Expand the Wizards directory.

Figure 10-1. Path to AWRConnected Mentor



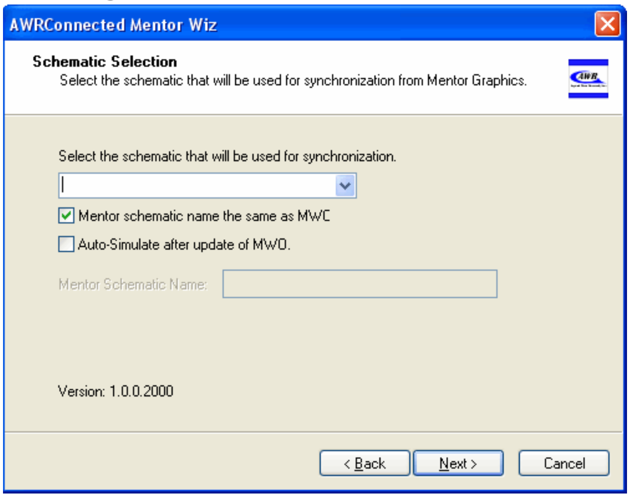
3. Choose **AWRConnected Mentor**. The AWRConnected Mentor Wiz window appears.

Figure 10-2. AWRConnected Banner



4. Click **Next** and the window in Figure 10-3 appears.

Figure 10-3. Schematic Selection

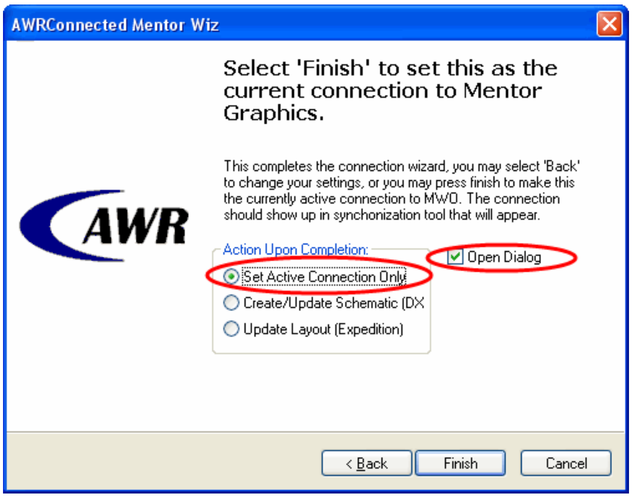


5. Select a schematic from the dropdown list.
6. Click **Next** and Figure 10-4 appears.

   Select one of the following:

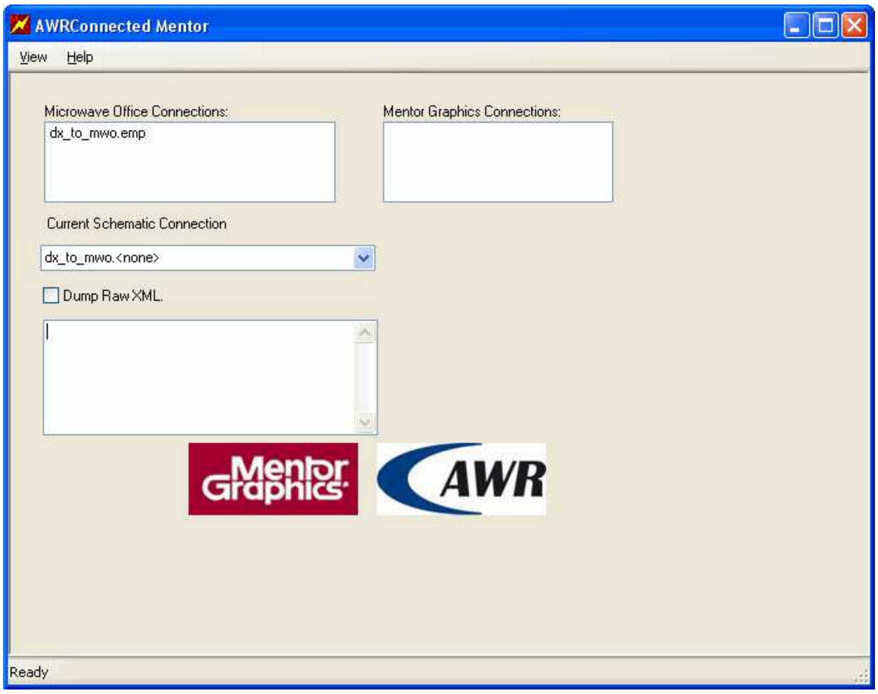
   - **Set Action Connection Only** — Launches only the AWR/MWO server.
   - **Create/Update Schematic (DX)** — Transfers design data from AWR to the schematic capture tool.
   - **Update Layout (Expedition)** — Transfers design data from AWR to the layout tool.

Figure 10-4. Current Connection



7. Click **Finish** and Figure 10-5 appears.

Figure 10-5. AWRConnected Mentor Dialog Box



8. Connect the schematic capture and/or layout tool to the AWR/MWO server with the following:
   - Connecting Schematic Capture to RF Simulator Server
   - Connecting Layout to RF Simulator Server

Results

You are now ready to connect the schematic capture and/or layout tool to the AWR/MWO server.

- Connecting Schematic Capture to RF Simulator Server
- Connecting Layout to RF Simulator Server

Related Topics

Schematic Capture

RF Layout

RF Simulator

Send Design Data to MWO Simulator

You can send and receive design data between the MWO RF simulator and the schematic capture or layout tools with the dynamic-link.

- Sending Schematic Capture RF Shapes
- Sending an MWO Design to Schematic Capture
- Sending Layout Data
- Posting Back MWO Updates to Layout

Related Topics

Schematic Capture
RF Layout
RF Simulator

Sending Schematic Capture RF Shapes

Send schematic capture RF shapes to the MWO schematic tool and create your design.

Use the following to integrate schematic capture AWR RF library symbols into AWR/MWO library.

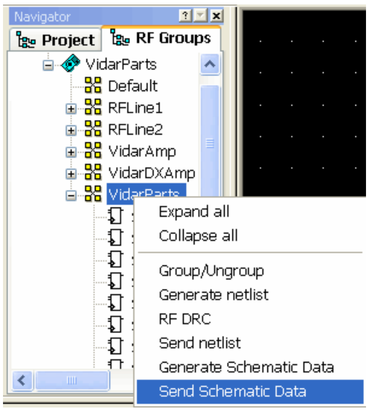
Prerequisites

- The schematic capture tool must be connected to the AWR/MWO server (Connecting Schematic Capture to RF Simulator Server).
- A schematic must be open in the AWR/MWO tool.

Procedure

1. Open a schematic in the Mentor Graphics' schematic capture tool.
2. Select the RF shapes and create an RF Group (Creating an RF Group).
3. Click the RF Groups tab in the **Navigator** (Figure 10-6).

Figure 10-6. Navigator, RF Groups Tab - RF Group Dropdown Menu



4. Select an RF group and choose **Send Schematic Data** from the popup.

Caution

Connect parts with a net when you create your design in the MWO schematic tool. A part-to-part connection is *not* supported in the schematic capture tool (RF Simulator Design Generation).

Result

The Mentor Graphic schematic capture RF shapes display in the RF simulator schematic.

Related Topics

Send Design Data to MWO Simulator

RF Simulator

Sending an MWO Design to Schematic Capture

After simulating your design and making design changes, update the design in the schematic capture tool.

Use the following procedure to send the MWO RF simulator design to the schematic capture tool.

Prerequisites

- The AWR/MWO RF simulator must be launched with the schematic opened.
- The schematic capture tool must be connected to the AWR server (Connecting Schematic Capture to RF Simulator Server).
- The schematic capture schematic name must be the same as the MWO schematic name.

Procedure

1. Launch the AWR/MWO RF simulator and open the schematic.
2. Select **Project Browser** > **Wizards** > **AWRConnected Mentor**.
3. Click **Next** in the AWRConnected Mentor Wiz dialog box.
4. Select **Create/update Schematic (DX)** and **Open Dialog** check box.
5. Click **Next**.
6. Select the schematic.
7. Click **Next**.
8. Click **Finish**.

Results

The MWO design appears in the schematic capture tool.

Related Topics

Send Design Data to MWO Simulator   RF Simulator

Sending Layout Data

Use the following procedure to send a layout RF group to the AWR/MWO RF simulator.

Prerequisites

- The layout tool must be connected the RF simulator (Connecting Layout to RF Simulator Server)
- A design must be open in the AWR/MWO layout tool.

Procedure

1. Open a design in the AWR/MWO layout tool.
2. Click **Control pane** ( ) or **Group/Ungroup** ( ) (RF Toolkit Toolbar).
3. Select an RF group in the Navigator.
4. Choose **Send layout data** (RF Group Menu Selections).

Results

If the RF group exits in the AWR/MWO layout tool, the RF shapes parameters, stackup, and locations of all elements in the RF group are updated.

The layout RF group appears in the AWR/MWO layout tool.

Related Topics

Send Design Data to MWO Simulator   RF Simulator

Posting Back MWO Updates to Layout

When you make design changes in the AWR/MWO RF simulator you can send them back to update the layout design.

Use the following procedure to send an AWR/MWO design back to the layout tool.

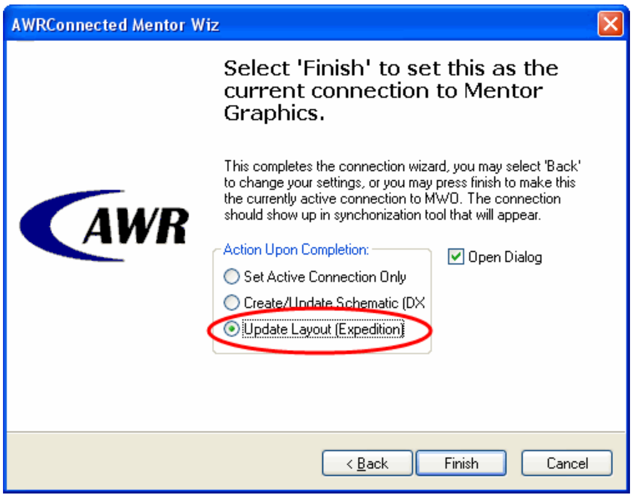
Prerequisites

- The layout tool must be connected to an AWR/MWO server (Connecting Layout to RF Simulator Server).

Procedure

1. Open your design in the layout tool.
2. Open your design in the AWR/MWO RF simulator.
3. Choose **View > Project Browser > Wizards > AWRConnected Mentor**.
4. Select **Update Layout (Expedition)** (Figure 10-7).

Figure 10-7. AWR Design to Layout



5. Click **Finish**.

Results

The AWR/MWO layout design displays in the layout tool.

Related Topics

Send Design Data to MWO Simulator   RF Simulator

Appendix A
Design Restrictions

The following design restrictions are known limitations.

- The layout tool does not allow you to delete schematic driven RF shapes.
- ADS design parameters that control the physical appearance of elements in the sub circuit are not supported.
- The RF elements RIBBON and WIRE are not currently supported in the ADS integration. The RF elements, MLCLE, MLVIAHOLE, and MLVIAPAD are not currently supported in ADS.
- The ADS RF simulator schematic capture tool and the layout tool use the same design unit.
- You must specify a seed component when placing RF parts in the layout tool.
- If you use any substrate other than the default, you must manually set up and maintain identical copies of the substrate in both DxDesigner and ADS.
- You cannot use the standard PCB reuse capabilities with RF designs.
- You cannot change nets assigned to RF shapes if the shape was created in the schematic tool. You can only assign existing nets to meanders and RF shapes if you create the shapes in the layout tool.
- AWR/MWO does not support connections with a net name in the Mentor Graphics schematic capture tool. The nets disconnect when you transfer the design to AWR/MWO schematic capture.

  Components have to be graphically connected and there is no support for connections between sheets. This restriction applies only to the AWR/MWO simulator.

RF Simulator Design Generation

When you create a design in the RF simulator, connect all objects with a net. Connecting objects with a net ensures object connection in the schematic capture tool. Refer to Figure A-1 and Figure A-2 for examples.

Figure A-1. Object to Net Connection

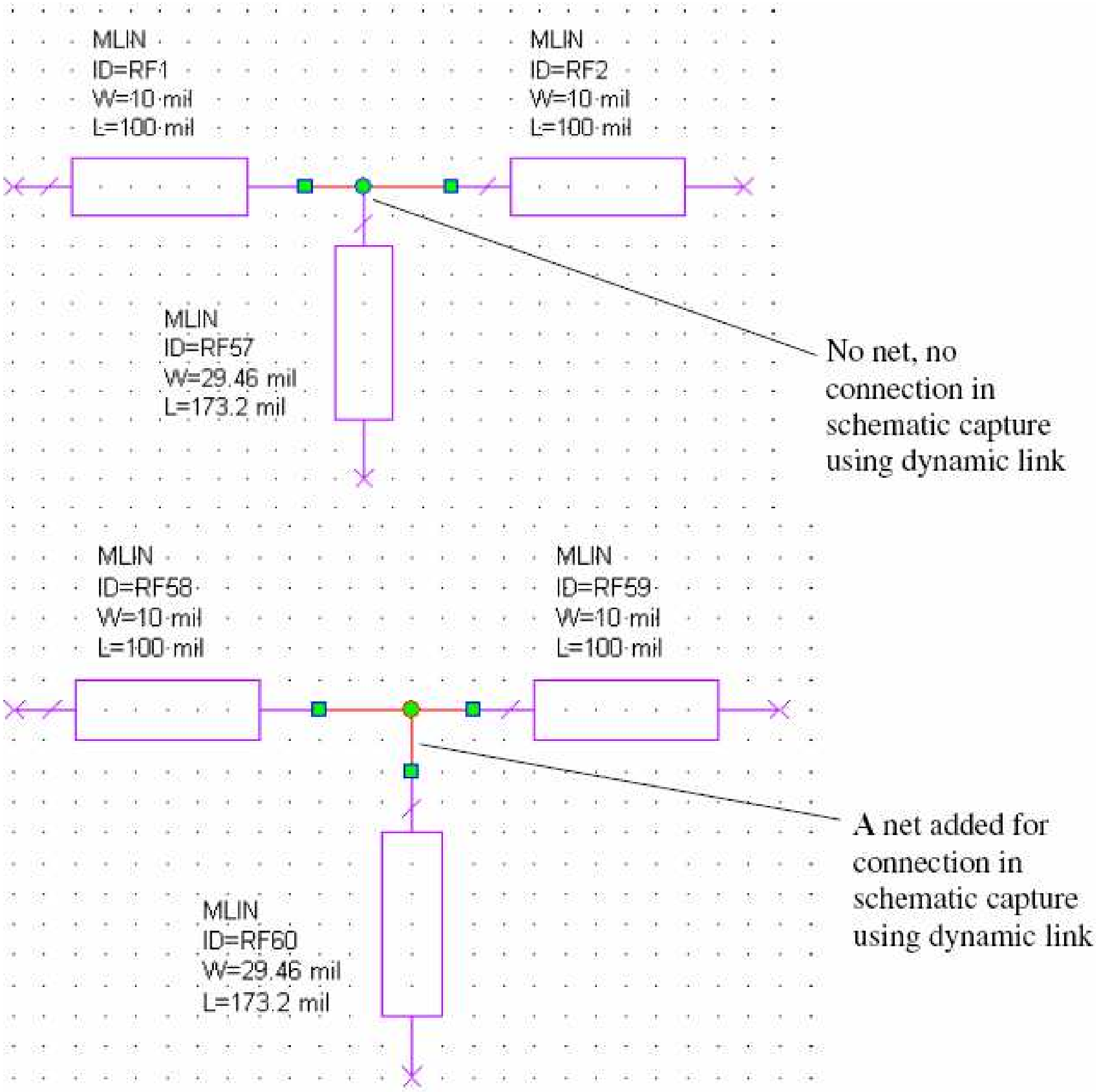
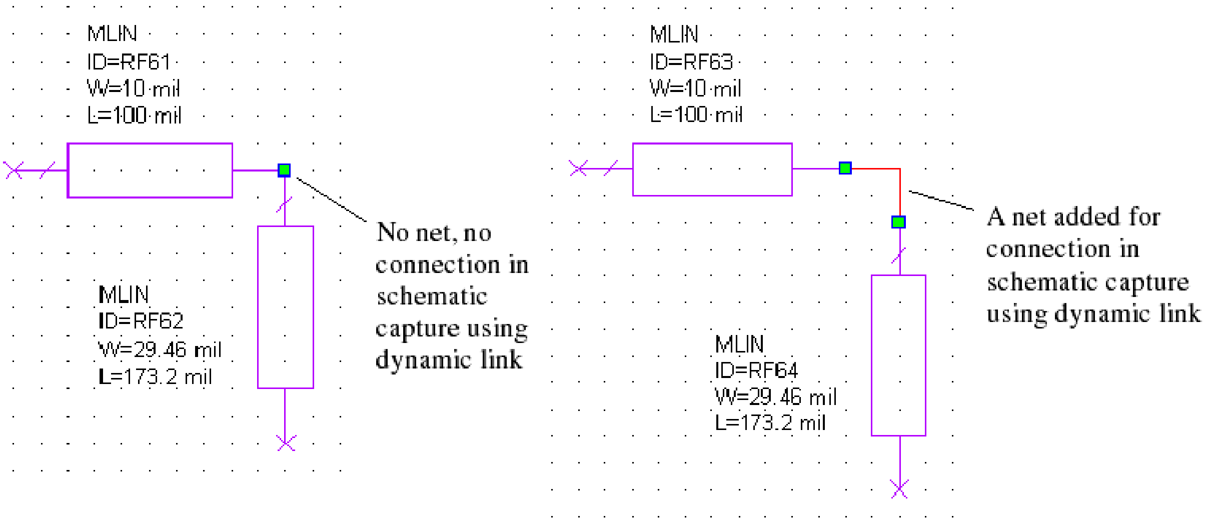
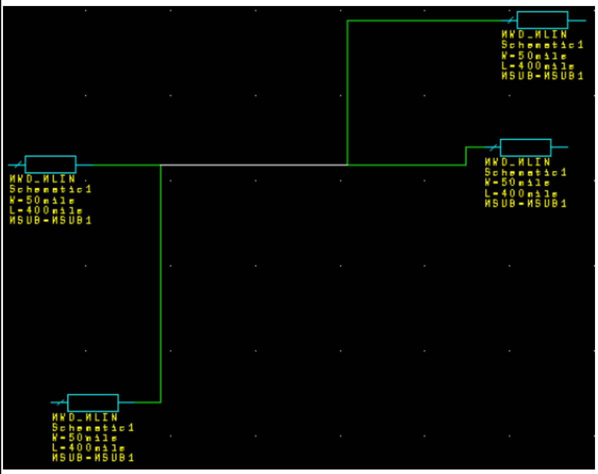
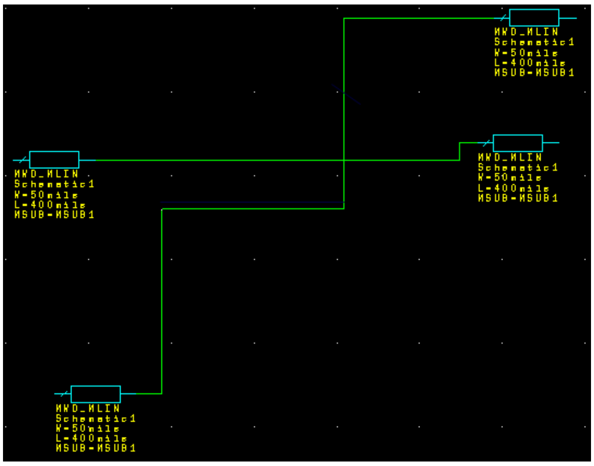


Figure A-2. Object to Object Connection



When you create a design in DxDesigner, make sure the nets or net segments **do not** overlay as shown in Figure A-3 — A. The RF simulator connects the overlaid nets or net segments. Create the DxDesigner design as shown in Figure A-3 — B.

Figure A-3. DxDesigner Nets

A - Segment overlay (Error)	B - No segment overlay (Correct)
	

Appendix B
Equation Editor Expressions

Table B-1 and Table B-2 list the supported functions for the Equation Editor Dialog Box.

Scaling information is available in Scale Factor.

The schematic capture tool and the layout tool support the following single argument functions.

Table B-1. Equation Editor Single Arguments List

Argument	Description
abs()	Returns the absolute value of the contents of the parentheses
acosh()	Returns the inverse hyperbolic cosine
arctan()	Returns the arc tangent of arg in radians; arctan() is the complementary function of tan().
atan()	Returns the inverse tangent, or arc tangent of arg in radians.
ceil()	Returns the next highest integer value rounded up if required.
cos()	Returns the cosine of arg in radians.
cosh()	Returns the inverse hyperbolic cosine or arg; the value whose hyperbolic cosine is arg.
cot()	Returns the cotangent of arg.
coth()	Returns the hyperbolic cotangent of arg.
ctof()	Converts Celsius to Fahrenheit returning a real number.
ctok()	Converts Celsius to Kelvin returning a real number.
db()	Returns the decibel measure of a voltage ratio of arg.
dbmtow()	Converts arg in dBm to watts.
deg()	Converts arg in radians to degrees.
exp()	Calculates the exponent of e to the power of arg.
floor()	Returns the largest integer lowest value by rounding down arg if necessary
ftoc()	Converts arg in Fahrenheit to Celsius and returns a real number.
ftok()	Converts Fahrenheit to Kelvin and returns a real number
int()	Converts real number arg into an integer.

Table B-1. Equation Editor Single Arguments List

ktoc()	Converts arg in Kelvin to Celsius as real number.
ktof()	Converts arg in Kelvin to Fahrenheit as real number.
ln()	Returns natural logarithm of arg.
log()	Returns the natural logarithm of base 10 as an integer or real number.
log10()	Calculates x (arg1) raised to the power of y (arg2).
mag()	Returns the magnitude of arg.
rad()	Converts arg in degrees to radians.
sin()	Returns the sine of arg in radians.
sinc()	Computes the sine of arg divided by arg (sin(arg)/arg), where arg is in radians.
sinh()	Computes the hyperbolic sine of arg.
sqrt()	Returns the square root of arg.
tan()	Computes the tangent of arg, where arg is in radians.
tanh()	Computes the hyperbolic tangent of arg in radians.
wtodbm()	Converts arg in Watts and returns the dBm as a real or complex number.

The schematic capture tool and the layout tool support the following double argument functions.

Table B-2. Equation Editor Double Arguments List

Argument	Definition
atan2()	Returns the inverse tangent, or arc tangent of arg which is the rectangular coordinates y and x.
dbm()	Returns the decibel measure of a voltage referenced to a 1 milliwatt signal and a defined impedance.
dbmtoa()	Converts arg's into short circuit current, given the reference impedance and returns a real or complex number.
dbmtov()	Converts arg's into open circuit voltage, given the reference impedance and returns a real or complex number.
fmod()	Returns the floating point remainder (modulo) of dividing arg1 by arg2
hypot()	Calculates the hypotenuse of a right triangle with side 1 (arg1) and side 2 (arg2).
jn()	Computes the Bessel function of the first kind and returns a real number of arg.
pow()	Calculates x (arg1) raised to the power of y (arg2).

Table B-2. Equation Editor Double Arguments List

rem()	Returns the remainder of arg1 divided by arg2 as a real number.
sgn()	Returns the integer sign of an integer or real number, as either 1 or -1.

abs()

Description

Returns the absolute value of the contents of the parentheses

Usage

p = abs(arg)

acosh()

Description

Returns the inverse hyperbolic cosine

Usage

p = acosh(arg)

arctan()

Description

Returns the arc tangent of *arg* in radians; **arctan()** is the complementary function of **tan()**.

Usage

p = arctan(arg)

atan()

Description

Returns the inverse tangent, or arc tangent of *arg* in radians.

Usage

p = atan(arg)

atan2()

Description

Returns the inverse tangent, or arc tangent of *arg* which is the rectangular coordinates y and x.

Usage

p = atan2(arg1, arg2)

ceil()

Description

Returns the next highest integer value rounded up if required.

Usage

p = ceil(arg)

cos()

Description

Returns the cosine of *arg* in radians.

Usage

p = cos(arg)

cosh()

Description

Returns the inverse hyperbolic cosine or *arg*; the value whose hyperbolic cosine is *arg*.

Usage

p = cosh(arg)

cot()

Description

Returns the cotangent of *arg*.

Usage

p = cot(arg)

coth()

Description

Returns the hyperbolic cotangent of *arg*.

Usage

p = coth(arg)

ctof()

Description

Converts Celsius to Fahrenheit returning a real number.

Usage

p = ctof(arg)

ctok()

Description

Converts Celsius to Kelvin returning a real number.

Usage

p = ctok(arg)

db()

Description

Returns the decibel measure of a voltage ratio of *arg*.

Usage

p = db(arg)

dbm()

Description

Returns the decibel measure of a voltage referenced to a 1 milliwatt signal and a defined impedance.

Usage

p = dbm(arg1, arg2)

*arg*1 — voltage (peak voltage)

*arg*2 — impedance

dbmtoa()

Description

Converts *arg's* into short circuit current, given the reference impedance and returns a real or complex number.

Usage

p = dbmtoa(arg1, arg 2)

arg1 — Value in dBm

arg2 — Reference Impedance

dbmtov()

Description

Converts *arg's* into open circuit voltage, given the reference impedance and returns a real or complex number.

Usage

p = dbmtov(arg1, arg2)

arg1 — Value in dBm

arg2 — Reference Impedance

dbmtow()

Description

Converts *arg* in dBm to watts.

Usage

p = dbmtow(arg)

deg()

Description

Converts *arg* in radians to degrees.

Usage

p = deg(arg)

exp()

Description

Calculates the exponent of **e** to the power of *arg*.

Usage

p = exp(arg)

floor()

Description

Returns the largest integer lowest value by rounding down *arg* if necessary

Usage

p = floor(arg)

fmod()

Description

Returns the floating point remainder (modulo) of dividing *arg1* by *arg2*

Usage

p = fmod(arg1, arg2)

ftoc()

Description

Converts *arg* in Fahrenheit to Celsius and returns a real number.

Usage

p = ftoc(arg)

ftok()

Description

Converts Fahrenheit to Kelvin and returns a real number

Usage

p = ftok(arg)

hypot()

Description

Calculates the hypotenuse of a right triangle with side 1 (arg1) and side 2 (arg2).

Usage

p = hypot(arg1, arg2)

int()

Description

Converts real number *arg* into an integer.

Usage

p = int(arg)

jn()

Description

Computes the Bessel function of the first kind and returns a real number of *arg*.

Usage

p = jn(arg1, arg2)

Set Type

arg1 — order

arg2 — the value for which the Bessel value is to be found

ktoc()

Description

Converts *arg in* Kelvin to Celsius as real number.

Usage

p = ktoc(arg)

ktof()

Description

Converts *arg in* Kelvin to Fahrenheit as real number.

Usage

p = ktof(arg)

ln()

Description

Returns natural logarithm of *arg*.

Usage

p = ln(arg)

log()

Description

Returns the natural logarithm of base 10 as an integer or real number.

Usage

p = log(arg)

log10()

Description

Returns the base-10 logarithm of *arg*.

Usage

p = log10(arg)

mag()

Description

Returns the magnitude of *arg*.

Usage

p = mag(arg)

pow()

Description

Calculates x (*arg1*) raised to the power of y (*arg2*).

Usage

p = pow(arg1, arg2)

arg1 — real number

arg2 — exponent

rad()

Description

Converts *arg* in degrees to radians.

Usage

p = rad(arg)

rem()

Description

Returns the remainder of *arg1 divided by arg2* as a real number.

Usage

p = rem(arg1, arg2)

sgn()

Description

Returns the integer sign of an integer or real number, as either 1 or -1.

Usage

p = sgn(arg)

sin()

Description

Returns the sine of *arg* in radians.

Usage

$p = sin(arg)$

sinc()

Description

Computes the sine of *arg* divided by *arg* (sin(*arg*)/*arg*), where *arg* is in radians.

Usage

p = sinc(arg)

sinh()

Description

Computes the hyperbolic sine of *arg*.

Usage

p = sinh(arg)

sqrt()

Description

Returns the square root of *arg*.

Usage

p = sqrt(arg)

tan()

Description

Computes the tangent of *arg*, where *arg* is in radians.

Usage

p = tan(arg)

tanh()

Description

Computes the hyperbolic tangent of *arg* in radians.

Usage

p = tanh(arg)

wtodbm()

Description

Converts *arg* in Watts and returns the dBm as a real or complex number.

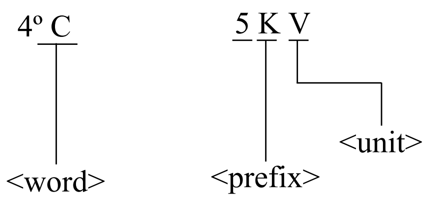
Usage

p = wtodbm(arg)

Scale Factor

A scale factor is a number which scales, or multiplies a fundamental unit. You can use the scale factor to define a parameter. A scale factor <*word*> begins with a letter or an underscore character (_). The remaining characters, if any, are the <*unit*><*prefix*> values consisting of letters, digits, and underscores.Values used for conversion from one unit of measure to another are scale factors.

Examples:



The <*word*> must match one of the words listed in Table B-3.

Table B-3. Word Scale Factor

Word	Numerical Equivalent	Definition
mil	$2.54*10^{-5}$	mils
in	$2.54*10^{-2}$	inches
ft	$12*2.54*10^{-2}$	feet
mi	$5280*12*2.54*10^{-2}$	miles
cm	$1.0*10^{-2}$	centimeters
mm	$1.0*10^{-3}$	millimeters
PHz	$1.0*10^{15}$	petahertz
dB	1.0	decibels
nmi	1852	nautical miles
C	1.0	Celsius
Prefixes are not allowed for mils, in, ft, mi, nmi.		

A set of fundamental units is a set of units used to describe measurement quantities which are equal to one. The case sensitive fundamental <*unit*>, must match the Unit listed in Table B-4.

Table B-4. Unit Scale Factor

Unit	Definition
A	Amperes
F	Farads

Table B-4. Unit Scale Factor (cont.)

Unit	Definition
H	Henries
Hz	Hertz
meter meters metre metres	meters
Ohm Ohms	Ohms
S	Siemens
sec.	seconds
V	Volts
W	Watts

The fundamental scale factors <*prefix*> must match one of the prefixes listed in Table B-4.

Table B-5. Prefix Scale Factor

Prefix	Numerical Equivalent	Definition
T	10^{12}	Tera
G	10^{9}	Giga
M	10^{6}	Mega
K	10^{3}	kilo
k	10^{3}	kilo
_ (underscore)	1	(no scale)
m	10^{-3}	milli
u	10^{-6}	micro
n	10^{-9}	nano
p	10^{-12}	pico
f	10^{-15}	femto
a	10^{-18}	atto

Scale factor resolution, within the Equation Editor, is evaluated in the following sequence <*word*> (Table B-3) <*unit*> (Table B-4) <*prefix*> (Table B-5). If the scale factors are not recognized, an error message is posted and a scale factor of 1 is used.

Table B-6. Operators

+	Addition
-	Subtraction

Table B-6. Operators (cont.)

*	Multiplication
/	Division
(	Open parenthesis
)	Close parenthesis
,	Comma - separator for arguments
=	Assignment
E	Exponent (as in 23.45E6

Equation expressions are evaluated, left to right. If parentheses are used, operation within the parentheses occurs first.

The mathematical operators in order of precedence are:

Table B-7. Mathematical Operators

^	Exponentiation
*, /	Multiplication and division
+, -	Addition and Subtraction

Related Topics

Auto Arranger

Var Block

End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:
www.mentor.com/eula

IMPORTANT INFORMATION

USE OF ALL SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE PRODUCTS. USE OF SOFTWARE INDICATES CUSTOMER'S COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.

END-USER LICENSE AGREEMENT ("Agreement")

This is a legal agreement concerning the use of Software (as defined in Section 2) and hardware (collectively "Products") between the company acquiring the Products ("Customer"), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity ("Mentor Graphics"). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, in the case of Software received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

1. **ORDERS, FEES AND PAYMENT.**

1.1. To the extent Customer (or if agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement ("Order(s)"), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not these documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order will not be effective unless agreed in writing by an authorized representative of Customer and Mentor Graphics.

1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will state separately in the applicable invoice(s). Unless timely provided with a valid certificate of exemption or other evidence that items are not taxable, Mentor Graphics will invoice Customer for all applicable taxes including, but not limited to, VAT, GST, sales tax and service tax. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. If Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under Orders placed by such third party in the event of default.

1.3. All Products are delivered FCA factory (Incoterms 2000), freight prepaid and invoiced to Customer, except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all Products delivered under this Agreement, to secure payment of the purchase price of such Products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation and design data ("Software") are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form (except as provided in Subsection 5.2); (b) for Customer's internal business purposes; (c) for the term of the license; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer requests any change or enhancement to Software, whether in the course of receiving support or consulting services, evaluating Software, performing beta testing or otherwise, any inventions, product

improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

3. **ESC SOFTWARE.** If Customer purchases a license to use development or prototyping tools of Mentor Graphics' Embedded Software Channel ("ESC"), Mentor Graphics grants to Customer a nontransferable, nonexclusive license to reproduce and distribute executable files created using ESC compilers, including the ESC run-time libraries distributed with ESC C and C++ compiler Software that are linked into a composite program as an integral part of Customer's compiled computer program, provided that Customer distributes these files only in conjunction with Customer's compiled computer program. Mentor Graphics does NOT grant Customer any right to duplicate, incorporate or embed copies of Mentor Graphics' real-time operating systems or other embedded software products into Customer's products or applications without first signing or otherwise agreeing to a separate agreement with Mentor Graphics for such purpose.

4. **BETA CODE.**

4.1. Portions or all of certain Software may contain code for experimental testing and evaluation ("Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. This grant and Customer's use of the Beta Code shall not be construed as marketing or offering to sell a license to the Beta Code, which Mentor Graphics may choose not to release commercially in any form.

4.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.

4.3. Customer agrees to maintain Beta Code in confidence and shall restrict access to the Beta Code, including the methods and concepts utilized therein, solely to those employees and Customer location(s) authorized by Mentor Graphics to perform beta testing. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 4.3 shall survive termination of this Agreement.

5. **RESTRICTIONS ON USE.**

5.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Products available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Products and ensure that any person permitted access does not disclose or use it except as permitted by this Agreement. Customer shall give Mentor Graphics written notice of any unauthorized disclosure or use of the Products as soon as Customer learns or becomes aware of such unauthorized disclosure or use. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, reverse-compile, reverse-engineer or in any way derive any source code from Software. Log files, data files, rule files and script files generated by or for the Software (collectively "Files"), including without limitation files containing Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") which are Mentor Graphics' proprietary syntaxes for expressing process rules, constitute or include confidential information of Mentor Graphics. Customer may share Files with third parties, excluding Mentor Graphics competitors, provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Software or Files or allow their use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Software, or disclose to any third party the results of, or information pertaining to, any benchmark.

5.2. If any Software or portions thereof are provided in source code form, Customer will use the source code only to correct software errors and enhance or modify the Software for the authorized use. Customer shall not disclose or permit disclosure of source code, in whole or in part, including any of its methods or concepts, to anyone except Customer's employees or contractors, excluding Mentor Graphics competitors, with a need to know. Customer shall not copy or compile source code in any manner except to support this authorized use.

5.3. Customer may not assign this Agreement or the rights and duties under it, or relocate, sublicense or otherwise transfer the Products, whether by operation of law or otherwise ("Attempted Transfer"), without Mentor Graphics' prior written consent and payment of Mentor Graphics' then-current applicable relocation and/or transfer fees. Any Attempted Transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and/or the licenses granted under this Agreement. The terms of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.

5.4. The provisions of this Section 5 shall survive the termination of this Agreement.

6. **SUPPORT SERVICES.** To the extent Customer purchases support services, Mentor Graphics will provide Customer updates and technical support for the Products, at the Customer site(s) for which support is purchased, in accordance with Mentor Graphics' then current End-User Support Terms located at http://supportnet.mentor.com/about/legal/.

7. **AUTOMATIC CHECK FOR UPDATES; PRIVACY.** Technological measures in Software may communicate with servers of Mentor Graphics or its contractors for the purpose of checking for and notifying the user of updates and to ensure that the Software in use is licensed in compliance with this Agreement. Mentor Graphics will not collect any personally identifiable data in this process and will not disclose any data collected to any third party without the prior written consent of Customer, except to Mentor Graphics' outside attorneys or as may be required by a court of competent jurisdiction.

8. **LIMITED WARRANTY.**

8.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Products, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Products will meet Customer's requirements or that operation of Products will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under an Order and does not renew or reset, for example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Products have been subject to misuse, unauthorized modification or improper installation. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF THE PRODUCTS TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF THE PRODUCTS THAT DO NOT MEET THIS LIMITED WARRANTY, PROVIDED CUSTOMER HAS OTHERWISE COMPLIED WITH THIS AGREEMENT. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) PRODUCTS PROVIDED AT NO CHARGE; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."

8.2. THE WARRANTIES SET FORTH IN THIS SECTION 8 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO PRODUCTS PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.

9. **LIMITATION OF LIABILITY.** EXCEPT WHERE THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT RECEIVED FROM CUSTOMER FOR THE HARDWARE, SOFTWARE LICENSE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 9 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

10. **HAZARDOUS APPLICATIONS.** CUSTOMER ACKNOWLEDGES IT IS SOLELY RESPONSIBLE FOR TESTING ITS PRODUCTS USED IN APPLICATIONS WHERE THE FAILURE OR INACCURACY OF ITS PRODUCTS MIGHT RESULT IN DEATH OR PERSONAL INJURY ("HAZARDOUS APPLICATIONS"). NEITHER MENTOR GRAPHICS NOR ITS LICENSORS SHALL BE LIABLE FOR ANY DAMAGES RESULTING FROM OR IN CONNECTION WITH THE USE OF MENTOR GRAPHICS PRODUCTS IN OR FOR HAZARDOUS APPLICATIONS. THE PROVISIONS OF THIS SECTION 10 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

11. **INDEMNIFICATION.** CUSTOMER AGREES TO INDEMNIFY AND HOLD HARMLESS MENTOR GRAPHICS AND ITS LICENSORS FROM ANY CLAIMS, LOSS, COST, DAMAGE, EXPENSE OR LIABILITY, INCLUDING ATTORNEYS' FEES, ARISING OUT OF OR IN CONNECTION WITH THE USE OF PRODUCTS AS DESCRIBED IN SECTION 10. THE PROVISIONS OF THIS SECTION 11 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.

12. **INFRINGEMENT.**

12.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Product acquired by Customer hereunder infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay costs and damages finally awarded against Customer that are attributable to the action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.

12.2. If a claim is made under Subsection 12.1 Mentor Graphics may, at its option and expense, (a) replace or modify the Product so that it becomes noninfringing; (b) procure for Customer the right to continue using the Product; or (c) require the return of the Product and refund to Customer any purchase price or license fee paid, less a reasonable allowance for use.

12.3. Mentor Graphics has no liability to Customer if the action is based upon: (a) the combination of Software or hardware with any product not furnished by Mentor Graphics; (b) the modification of the Product other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of the Product as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code or Product provided at no charge; (g) any software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; or (h) infringement by Customer that is deemed willful. In the case of (h), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.

12.4. THIS SECTION 12 IS SUBJECT TO SECTION 9 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS FOR DEFENSE, SETTLEMENT AND DAMAGES, AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY, WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY PRODUCT PROVIDED UNDER THIS AGREEMENT.

13. **TERMINATION AND EFFECT OF TERMINATION.** If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term.

13.1. Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement immediately upon written notice if Customer: (a) exceeds the scope of the license or otherwise fails to comply with the licensing or confidentiality provisions of this Agreement, or (b) becomes insolvent, files a bankruptcy petition, institutes proceedings for liquidation or winding up or enters into an agreement to assign its assets for the benefit of creditors. For any other material breach of any provision of this Agreement, Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement upon 30 days written notice if Customer fails to cure the breach within the 30 day notice period. Termination of this Agreement or any license granted hereunder will not affect Customer's obligation to pay for Products shipped or licenses granted prior to the termination, which amounts shall be payable immediately upon the date of termination.

13.2. Upon termination of this Agreement, the rights and obligations of the parties shall cease except as expressly set forth in this Agreement. Upon termination, Customer shall ensure that all use of the affected Products ceases, and shall return hardware and either return to Mentor Graphics or destroy Software in Customer's possession, including all copies and documentation, and certify in writing to Mentor Graphics within ten business days of the termination date that Customer no longer possesses any of the affected Products or copies of Software in any form.

14. **EXPORT.** The Products provided hereunder are subject to regulation by local laws and United States government agencies, which prohibit export or diversion of certain products and information about the products to certain countries and certain persons. Customer agrees that it will not export Products in any manner without first obtaining all necessary approval from appropriate local and United States government agencies.

15. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. All Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to US FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in this Agreement, except for provisions which are contrary to applicable mandatory federal laws.

16. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.

17. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FLEXlm or FLEXnet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this Section 17 shall survive the termination of this Agreement.

18. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of certain Mentor Graphics intellectual property licensed under this Agreement are located in Ireland and the United States. To promote consistency around the world, disputes shall be resolved as follows: excluding conflict of laws rules, this Agreement shall be governed by and construed under the laws of the State of Oregon, USA, if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of the courts of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply. Notwithstanding the foregoing, all disputes in Asia arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the chairman of the Singapore International Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. This section shall not

restrict Mentor Graphics' right to bring an action against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.

19. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.

20. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements, including but not limited to any purchase order terms and conditions. Some Software may contain code distributed under a third party license agreement that may provide additional rights to Customer. Please see the applicable Software documentation for details. This Agreement may only be modified in writing by authorized representatives of the parties. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.

Rev. 100615, Part No. 246066