cādence®

X Commands

Product Version 23.1
September 2023

Contents

1

X Commands

xcomp	xdbgroup	xdbgrouptype
xdehilite	xdevsym	xdevtype
xdrawing select	xfindprop	xfunc
xhilite	xname_flush	xnet
xratbundle	xrefdes	xsection
xsection_chart	xsymbol	xsymtype
xymode		

xcomp

The `xcomp` command is used in conjunction with `property edit` to locate objects by component name, and with `show element` to display information on the named objects. It differs from the `comp` command in that the `xcomp` action is deferred until you run `xname_flush`. This allows you to find/choose multiple component instances.

Related Topics

- Displaying Component Information
- Choosing Objects for Property Editing Using the Xcomp Command

Xcomp Dialog Box

Depending on the command you run `xcomp` with, the following dialog boxes are displayed:

- Show Element
- Edit Property
- Show Properties

Related Topics

- Choosing Objects for Property Editing Using the Xcomp Command

Displaying Component Information

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose object type Comps in the Find filter.

3. Type `xcomp` <*component name*> at the console window prompt.

4. Repeat step 2 for each additional component instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.
   The Show Element display window for the specified components appears.

Related Topics

- xcomp

Choosing Objects for Property Editing Using the Xcomp Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the object type *Comps* in the *Find* filter.

3. Type `xcomp` <*component name*> at the console window prompt of your user interface.

4. Repeat step 2 for each component which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the components for the selected objects as described in property edit.

Related Topics

- xcomp
- Xcomp Dialog Box

xdbgroup

The `xdbgroup` command is used in conjunction with `show element` to display information on the named objects, and with certain Edit commands to locate objects by group name. It differs from the `dbgroup` command in that the `xdbgroup` action is deferred until you run `xname_flush`. This allows you to find/choose multiple groups.

Related Topics

- Displaying Information of Specified Groups

- Choosing Objects for Property Editing using the Xdbgroup Command

Xdbgroup Dialog Box

Depending on the command you run `xdbgroup` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Property Editing using the Xdbgroup Command

Displaying Information of Specified Groups

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose object type *Groups* in the *Find* filter.

3. Type `xdbgroup` <*group name*> at the console window prompt.

4. Repeat step 2 for each additional group instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.
   The Show Element display window for the specified groups appears.

Related Topics

- xdbgroup

Choosing Objects for Property Editing using the Xdbgroup Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the object type *Groups* in the *Find* filter.

3. Type `xcomp` <*group name*> at the console window prompt.

4. Repeat step 2 for each group which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the groups for the selected objects as described in property edit.

To choose objects for other Edit commands, run the appropriate supported command; for example, `copy`.

Related Topics

- xdbgroup
- Xdbgroup Dialog Box

xdbgrouptype

An internal Cadence engineering command.

xdehilite

The `xdehilite` command works in the same fashion as dehilight

xdevsym

The `xdevsym` command is used in conjunction with `property edit` to locate objects by device type, with `show element` to display information on the named objects, and on certain Edit commands. It differs from the `devsym` command in that the `xdevsym` action is deferred until you run `xname_flush`. This allows you to find/choose multiple device types.

> ⚠ This command is similar to `xdevtype`. The difference is in the type of information displayed in the Show Element dialog box when the command is run with `show element`.

Related Topics

- Displaying Information for Specified Device Symbols
- Choosing Objects for Property Editing using the Xdevsym Command

Xdevsym Dialog Boxes

Depending on the command you run `xdevsym` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Property Editing using the Xdevsym Command

Displaying Information for Specified Device Symbols

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose the appropriate object types in the *Find* filter.

3. Type `xdevtype` <*device type name*> at the console window prompt.

4. Repeat step 2 for each additional device type instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.
   The Show Element display window for the specified device types appears.

Related Topics

- xdevtype

- xdevsym

Choosing Objects for Property Editing using the Xdevsym Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the appropriate object types in the *Find* filter.

3. Type `xdevsym` *<device symbol name>* at the console window prompt.

4. Repeat step 2 for each device type which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the device symbols for the selected objects as described in property edit.
   To choose objects for other Edit commands, run the appropriate supported command; for example, `copy`.

Related Topics

- xdevsym
- Xdevsym Dialog Boxes

xdevtype

The `xdevtype` command is used in conjunction with `property edit` to locate objects by device type, with `show element` to display information on the named objects, and on certain Edit commands. It differs from the `devtype` command in that the `xdevtype` action is deferred until you run `xname_flush`. This allows you to find/choose multiple device types.

> ⚠ This command is similar to `xdevsym`. The difference is in the type of information displayed in the Show Element dialog box when the command is run with `show element`.

Related Topics

- Displaying Information for Specified Device Symbols
- Choosing Objects for Property Editing using the Xdevtype Command

Xdevtype Dialog Box

Depending on the command you run `xdevtype` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Property Editing using the Xdevtype Command

Choosing Objects for Property Editing using the Xdevtype Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the appropriate object types in the *Find* filter.

3. Type `xdevtype` <*device type name*> at the console window prompt.

4. Repeat step 2 for each device type which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the device types for the selected objects as described in property edit.
   To choose objects for other Edit commands, run the appropriate supported command; for example, `copy`.

Related Topics

- xdevtype
- Xdevtype Dialog Box

xdrawing select

The `xdrawing select` command works in a similar fashion as drawing select. The command allows you to select your entire active design in conjunction with another command; for example `property edit`.

Selecting Your Design for Use with a Command

To select your design using a command, perform the following steps:

1. Run the `property edit` command.

2. Choose the appropriate object types in the *Find* filter.

3. Type `xdrawing select` at the console window prompt.

4. Type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

5. Edit the object types for the selected objects as described in property edit.

xfindprop

The `xfindprop` command is used in conjunction with `property edit` to locate objects by property, and with `show element` to display information on the named objects. It differs from the `findprop` command in that the `xfindprop` action is deferred until you run `xname_flush`. This allows you to find/choose multiple property instances.

Related Topics

- Displaying Information for Specified Properties
- Choosing Objects for Property Editing using the Xfindprop Command

Xfindprop Dialog Boxes

Depending on the command you run `xfindprop` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Property Editing using the Xfindprop Command

Displaying Information for Specified Properties

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose the appropriate object types in the *Find* filter.

3. Type `xfindprop` <*property name*> at the console window prompt.

4. Repeat step 2 for each additional property instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.
   The Show Element display window for the specified properties appears.

Related Topics

- xfindprop

Choosing Objects for Property Editing using the Xfindprop Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the appropriate object types in the *Find* filter.

3. Type `xfindprop` <*property name*> at the console window prompt.

4. Repeat step 2 for each property which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the properties for the selected objects as described in property edit.

Related Topics

- xfindprop
- Xfindprop Dialog Boxes

xfunc

The `xfunc` command is used in conjunction with `property edit` to locate objects by function instance, and with `show element` to display information on the named objects. It differs from the `func` command in that the `xfunc` action is deferred until you run `xname_flush`. This allows you to find/choose multiple function instances.

Related Topics

- Displaying Information for Specified Function Instances
- Choosing Objects for Property Editing using the Xfunc Command

Xfunc Dialog Boxes

Depending on the command you run `xfunc` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Property Editing using the Xfunc Command

Displaying Information for Specified Function Instances

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose object type *Functions* in the *Find* filter.

3. Type `xfunc` <*function designator name*> at the console window prompt.

4. Repeat step 2 for each additional function instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.

The Show Element display window for the specified function instances appears.

Related Topics

- xfunc

Choosing Objects for Property Editing using the Xfunc Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose object type *Functions* in the *Find* filter.

3. Type `xfunc` <*function designator name*> at the console window prompt.

4. Repeat step 2 for each additional function instance which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the properties for the selected functions as described in property edit.

Related Topics

- xfunc
- Xfunc Dialog Boxes

xhilite

The `xhilite` command works in a similar fashion to the hilight command.

xname_flush

The xname_flush command is used to complete the actions of the following commands:

- xcomp

- xdbgroup

- xdevsym

- xdevtype

- xfindprop

- xfunc

- xnet

- xrefdes

- xsymbol

- xsymtype

xnet

The `xnet` command is used in conjunction with `property edit` to locate nets, and with `show element` to display information on the named selections. It differs from the `net` command in that the `xnet` action is deferred until you run `xname_flush`. This allows you to find/select multiple nets.

Related Topics

- Displaying Information for Specified Nets
- Choosing Objects for Property Editing using the Xnet Command

Xnet Dialog Boxes

Depending on the command you run `xnet` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Property Editing using the Xnet Command

Displaying Information for Specified Nets

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Type `xnet` <*net name*> at the console window prompt.

3. Repeat step 2 for each additional function instance on which you want information.

4. When done, type `xname_flush` at the console window prompt.

The Show Element display window for the specified nets appears.

Related Topics

- xnet

Choosing Objects for Property Editing using the Xnet Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose object type *Nets* in the *Find* filter.

3. Type `xnet` *<net name>* at the console window prompt.

4. Repeat step 2 for each additional net which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the properties for the selected functions as described in property edit.

Related Topics

- xnet
- Xnet Dialog Boxes

xratbundle

The x_{ratbundle} command is used to select multiple bundles in the design by name via the command line. It can be used several times directly before or after certain commands that operate on the named bundles. For example, `show element` to display information about the named bundles.

> ⚠ The action on the bundles is deferred until you run the `xname_flush` command. This enables you to select multiple bundles.

Syntax

```
xratbundle <bundle_name>
```

Example

```
xratbundle <bundle_name>...xname_flush
```

Displaying Information About Named Bundles

Perform the following steps to display information about named bundles:

1. In the console window, type `xratbundle` followed by the name of a bundle in the design. For example:

   `xratbundle bndl_5`

   The bundle highlights.

2. Repeat step 1, until all desired bundles are selected.

3. In the console window, type `xname_flush` to capture all the bundle names for the next command.

4. Run the `show element` command.

   Information about the named bundles is displayed in the Show Element window.

xrefdes

The `xrefdes` command is used in conjunction with an active command, such as `place manual`, `property edit`, and certain Edit commands. It lets you find/choose components when you type in the command followed by the objects reference designators. It differs from the `refdes` command in that the `xrefdes` action is deferred until you run `xname_flush`. This allows you to find/choose multiple reference designators.

Related Topics

- Displaying Information for Specified Reference Designators
- Choosing Objects for Editing using the Xrefdes Command

Xrefdes Dialog Boxes

Depending on the command you run `xnet` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Editing using the Xrefdes Command

Displaying Information for Specified Reference Designators

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Type `xrefdes` *<refdes name>* at the console window prompt.

3. Repeat step 2 for each additional reference designator on which you want information.

4. When done, type `xname_flush` at the console window prompt.

The Show Element display window for the specified reference designators appears.

Related Topics

- xrefdes

Choosing Objects for Editing using the Xrefdes Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. choose the appropriate object types in the *Find* filter.

3. Type `xrefdes` <*refdes name*> at the console window prompt.

4. Repeat step 2 for each additional reference designator which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the properties for the selected functions as described in property edit.
   To choose objects for other Edit commands, run the appropriate supported command; for example, `copy`.

Related Topics

- xrefdes
- Xrefdes Dialog Boxes

xsection

The `xsection` command displays Cross-section Editor and provides a setup to let you view and edit the layout cross-section, to set dynamic unused pad suppression, and to set up parameters for embedded component design.

The default view of Cross-section Editor combines the spreadsheet grid with the cross-section viewer. The drill display within the viewer is based on actual padstack usage in the database. and do not display unused vias in Physical CSets.
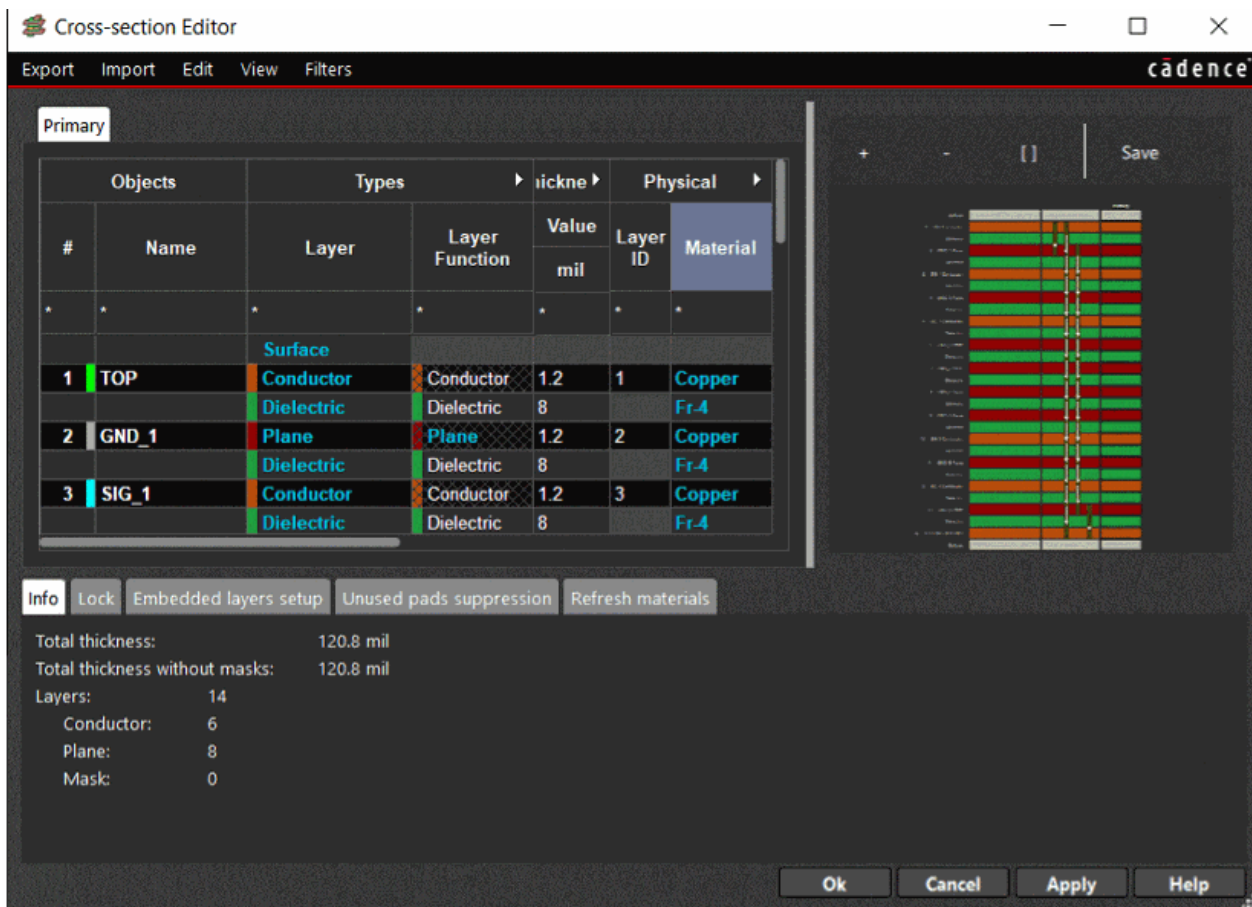
The cross-section consists of the ordered layers of your design, including information about their type, thickness, spacing, electrical characteristics, and impedance. The Cross-section Editor worksheet presents materials in groups, as conductive and non-conductive. The determination of *Type* is made based upon the material's electrical conductivity (E_CONDUCTIVITY) according to the following rule:

E_CONDUCTIVITY > 100,000 mhos/m = conductive

E_CONDUCTIVITY < 100,000 mhos/m = non-conductive

> ⚠ Semiconductor materials are not currently supported.

When Using SiP Tools and Die Stacks

When viewing the cross-section of a SiP design with die stacks, you will see that the substrate layers are red and the die layers are blue. Die layers are the layers you create for dies, spacers, and interposers. Information for the die layers is grayed out. You need to access the diestack editor to obtain or edit this information.

Related Topics

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Cross-Section Editor Dialog Box

Access Using

- Menu Path: *Setup – Cross-section*

- Toolbar Icon:

The cross-section worksheet presents the layers of the active design using a spreadsheet where rows represent the primary layer material and columns represent the various properties of the layer. You can resize the dialog box to display a larger range of layers in the design.

The Cross-section Editor dialog box automatically displays default values that are in material files (`materials.dat` or `mmcmmat.dat`). These files provided by the layout editor contain typical industry fabrication materials. They are located in directories specified in the search path defined by the `$MATERIALPATH` environment variable.

You can modify most attributes by entering a new value in the appropriate cell. Modified values are displayed in blue font with bold emphasis. While the values equivalent to the material file are displayed in black font with regular emphasis. You cannot modify attributes for the extreme outer layers that have a fixed name called SURFACE and no definable attributes, and the extreme outer CONDUCTOR layers, which have a fixed name of TOP and BOTTOM. You cannot change the name TOP and BOTTOM but you can change the values on those layers.

Related Topics

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Cross-Section Editor Dialog Box Controls

The Cross-Section Editor Dialog Box has the following options:

- Cross-Section Editor Menus

- Dialogs Pane

- Worksheet Controls

- Drill Chart Viewer

Cross-section Editor Menus

Export		
	Cross-section Technology File	Choose to export the cross-section information to the XML based technology file (`.tcfx`). The cross-section information includes all the conductor, surface, dielectric, and die stack layers and their characteristics.
	IPC2581	Choose to export the cross-section information to the XML based IPC2581 file (`.xml`). The cross-section information includes all the conductor, surface, dielectric, and die stack layers and their characteristics. ⚠ Manufacturing layer names are not exported to the XML file.
	HTML File	Choose to export the cross-section information to a HTML file. In Multi Stackups mode, the cross-section information of active stackup is exported. The cross-section information includes all the conductor, surface, dielectric, and die stack layers and their characteristics.
Import		
	Technology File	Choose to import a technology Constraint File `.tcfx` file into your design. For more information, see Import a technology file (.tcfx) Dialog Box.
	IPC2581	Choose to import an IPC2581 file `.xml` file into your design.
View	Supports functions to reset the grid and chart controls.	

	Multi Stackups mode	Show multiple tabs for different stackups.
	Show All Columns	Show all the columns in the spreadsheet.
	Show Float Cross-section Chart	Choose to undock the cross-section chart
	Show Docked Cross-section Chart	Choose to dock the cross-section chart
	Show Drill Chart	Choose to display a Drill Chart.
	Show Dialogs Pane	Choose to control the display of Dialogs pane.
	Cross-section Viewer Draw Options	Display *Draw options* dialog box.
	Options	Displays *UI Options* Dialog box.
	Reset UI to Cadence Default	Restore the default settings for Cross-section Editor
	Report	Choose to create Cross-Section Report in a design for each layer.
Filters		
	Show Only Conductor Layers	Choose to display only conducting layers.

	Reset all filters	Choose to reset all filters when only conducting layers are displayed.

Draw Options Dialog Box

The dialog box displays options for creating cross-section layers.

> ⚠ The layer and background gradient draw options works when OpenGL is supported and enabled.

UI Options Dialog Box

For more information on UI Options see, View Options Dialog Box in the Allegro Constraint Manager Reference.

Dialogs Pane

The five functional tabs are located near the bottom of the spreadsheet.

Info			
	Total thickness	Displays total thickness in database units.	
	Total thickness without masks	Displays total thickness in database units.	
	Layers	Displays total number of layers (Plane, Etch, and Mask).	
Lock	Prevents editing within spreadsheet		
	Add Layers	Locks context-sensitive layer editing options in the *Objects* column of the spreadsheet	
	Values change	Locks complete grid and made all the values read-only to prevent them for being modified	
Embedded Layers Setup	Displays setup form for Embedded Component Design. ⚠ This option is available only when Miniaturization option.		

	Package height buffer	Specifies the minimum gap to be maintained between the embedded component and the etch layer. For example, if the gap between two layers is 20 mils, and cavity to etch layer clearance is 3 mils, the maximum height of the component on this layer can only be 17 mils.
	Minimum cavity gap for merging	Specifies the minimum gap in the XY-direction that is to be maintained between two cavities before they can be merged.
	Placebound to via keepout expansion	Creates a via keepout outline from Placebound if library symbol does not have it.
	Package to cavity spacing	Specifies the minimum gap to be maintained in the XY-direction between the embedded component and the cavity surrounding it.
	Via connect height	This parameter is defined only when Indirect Attach method of component placement is used. It specifies the height of the vias used for connecting an embedded component to the etch layer. This value gets added to the PACKAGEHEIGHT to calculate effective package height.
	Default via connect padstack	This parameter is defined only when Indirect Attach method of component placement is used. Specifies the default padstack to be used if the EMB_VIA_CONNECT_PADSTACK property is not specified on the drawing.
	Cavity to route keepout expansion	This parameter is defined only when Protruding Allowed option is enabled for the etch layer. This indicates the minimum distance between a protruding cavity and the routes on the etch layer.
Unused Pads Suppression	Displays setup form for Unused Pads Suppression.	

	Dynamic unused pads suppression	When enabled, dynamically adds pads when a connection to a pin or via occurs, and removes them when the connection is deleted. If disabled, retains the per-layer settings for pins and vias as defined in the dialog box, but restores all suppressed pads globally, even if DRC errors result. Although enabling this option automatically enables *Display Padless Holes,* the converse is not true.
	Display padless holes	Displays padless holes for pins or vias whose visible pads are NULL or suppressed for visual guidance during etch editing. You can disable this option even if *Dynamic unused pads suppression* is enabled to obtain a true representation of artwork, which excludes padless holes. Conversely, disabling *Dynamic unused pads suppression* does not automatically disable this option. This option is also available on the *Display* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (prmed command). Changing this option in one location automatically updates the other as well. Enabling *Display Plated Holes* or *Display Non-plated Holes* also enables *Display Padless Holes*, which can be enabled even if *Display Plated Holes* and *Display Non-plated Holes* are disabled.
Refresh Materials		Refreshes the cross-section worksheet with changes that have been made in the Materials file (`materials.dat`) You can choose which properties to update by selecting from a menu containing the following options.
	All values	Refreshes all values in the worksheet that are associated with materials.
	Electrical Conductivity	Refreshes just the values in the *Conductivity* column.
	Dielectric Constant	Refreshes just the values in the *Dielectric Constant* column for dielectric layers only. Enable this option to update the dielectric values when a new material is added in the Material Editor.
	Loss Tangent	Refreshes just the values in the *Loss Tangent* column for dielectric layers only.
	Material Thickness	Refreshes just the values in the *Thickness* column.

	Frequency Dependent File	Refreshes all values in the Frequency Dependent File column.
	Refresh Materials	Refreshes all values in the worksheet that are associated with materials.

Worksheet Controls

There are two tabs *Physical* and *All*. The Physical tab provides a limited view of the Cross-section spreadsheet. Categories filtered out include Signal Integrity, Embedded Component Design and Properties.

> ⚠ Some of the controls described below may not be available in the Allegro product or tier you are working in.

Option	Description	
Objects		
	Name	Represents the name of the layer.
Types	Specifies the layer type.	
	Layer	Represents the name of the layer. • SURFACE • CONDUCTOR • DIELECTRIC • PLANE • DIE STACK

	Layer Function	Lists various type of the materials used to specify dielectric and mask layers. Some of the available options are: • Capacitive • Coating Conductive • Coating Non Conductive • Conductive Adhesive • Conductive Film • Conductive Foil • Dielectric Adhesive • Dielectric Base • Dielectric Coverlay • Dielectric Prepeg • Resistive • Silkscreen • Solder Paste • Solder Mask
	Manufacture	Assigns hierarchical names to signal and plane layers that align with IPC2581 data schema. This information is currently not exported to IPC2581 XML file. Use this column to specify user-defined manufacturing layer names for informational purposes when viewing a cross-section. For example, during fabrication sequential lamination of a board is done in multiple stages. Assigning manufacturing layer names help to identify different fabrication stages in Cross-section Editor.
	Constraint	Assigns hierarchical names to signal and plane layers similar to the *Manufacture* column. Their names are, however, integrated into Spacing CSet structures and contribute to the use of generic tech files.
Thickness		

	Value	Specifies the thickness value, which you can change, for the currently selected layer. The total thickness for all layers appears in the *Info* tab below the horizontal scroll bar of the Cross-Section worksheet. Be sure to define the proper thickness before doing any simulation or impedance calculation. To check the layer thickness, choose *Analyze – SI – Audit – Design Audit.*
	(+) Tol.	Specifies positive tolerance value.
	(-) Tol.	Specifies negative tolerance value.
Physical		
	Layer ID	Allows customization of BB via label display. The value supports up to three alpha-numeric characters.
	Material	You can choose from default materials as currently specified in your Materials file. Material choices are based on the selected layer type. For further information on default materials, see Default Cross Section Values in *Preparing the Layout* in your documentation set. The maximum character limit is increased from 19 to 250. Material names are stored in the `materials.dat` file for PCB and `mcmmat.dat` for packaging tools. ⚠ For *Dielectric* layers, changing material automatically updates *Layer Function* type.
	Fill-in Material	Specifies the fill-in material for the layer. Material choices are based on the selected layer type. By default, this field is enabled only for *Conductor* layers, and the fill-in material options provided are dielectric materials.
	Negative Artwork	When checked, creates negative artwork for the selected conductor layer.
	No Fillet	If enabled for a layer, prevents creation or regeneration of fillets on pins and vias across the entire layer. This option overrides the value of *Dynamic* fillet setting in the *Fillet and Tapered Trace* dialog box.

	Unused Pin Suppression	Controls the layer settings for the removal of unused pads on inner signal layers. As a result, fields are grayed out for Top/Bottom and Negative layers.
	Unused Via Suppression	Control the layer settings for the removal of unused pads on inner signal layers. As a result, fields are grayed out for Top/Bottom and Negative layers
Embedded	Supporting the setup of Embedded Component Design	

	Embedded Status	Specify whether the layer can be used for component placement and if used, the orientation of the component on the layer. The supported values are:
		
		• Not Embedded
		  Components cannot be placed in this layer
		
		• Body Up
		  The layer can be used for placement of packaged component; and the body of the component placed on this layer is oriented toward the Top surface of the PCB.
		
		• Body Down
		  The layer can be used for component placement, however, the body of the component is oriented toward the Bottom surface of the PCB.
		
		• Protruding Allowed
		  This enables the embedded component placed on the adjacent signal layers to cut across the current layer.
		  This option allows placement of embedded components for which package height is greater than the dielectric thickness between two layers.

	Attach Method	Specify the method to be used for connecting the components to the embedded layer. The options supported are:
		• Direct Attach The component is placed (soldered) directly to the etch layer.
		• Indirect Attach Component is not placed on the etch layer directly. It is suspended in the dielectric material and vias are used to connect the component and the etch layer.
Signal Integrity		
	Conductivity mho/cm	Specifies the electrical conductivity for the selected layer. Entries having a unit of measure other than mho/cm are converted to mho/cm.
	Dielectric Constant	Specifies the dielectric constant value for the selected layer.
	Width um	Defines the width of the routed etch line on the layer. The default is referenced from the active physical rule set. When a different unit of measure is entered in addition to the number, the value is converted to the currently selected unit of measure and the impedance is automatically recalculated to correspond to changes you make in the *Line Width* column. ⚠ Changes that you make to *Line Width*, in the cross-section worksheet, do not affect the line width values in the constraint set.
	Impedance Ohm	Sets the impedance of etch lines on the layer. The only unit of measure accepted in the *Impedance* column is ohms. The line width automatically recalculates to correspond to changes you make in the *Impedance* column.

	Loss Tangent	Specifies the dielectric losses for the currently selected layer in terms of the tangent of the complement of the insulation power-factor angle. The *Impedance* value changes when you modify the *Loss Tangent* value. ⚠ See your materials vendor for the actual value.
	Shield	Designates the currently selected plane layer as a shield layer. The shield layer prevents the electrical signals from two adjacent layers from interacting with each other. When *Shield* is checked, the simulator treats the layer as a pseudo-infinite reference plane for a transmission line and uses actual shape boundaries to determine the reference plane for a transmission line. For example, a trace is modeled as two transmission lines connected in series where it runs off the edge of a ground plane. The two transmission lines probably have different impedance values, because they have different reference plane spacing.
	Freq. Dep. File	Specifies the Frequency Dependent File selectable from the files residing in your MATERIALPATH directory, `//<install_directory/share/pcb/test/materials`. The frequency-dependent material file (`.material`) defines frequency-dependent materials containing electrical properties for individual materials (for example, copper) defined over a range of frequencies for the purpose of modeling the delay and dispersive behavior of arbitrary materials. For additional information, see Dispersive Dielectric Material Support in the *Allegro PCB SI User Guide.*
	Etch Factor	Enables you to define a layer-specific trapezoidal angle for each conductor and plane layer in your design. The default for all valid layers is 90 degrees. You can edit values in the range of 45 - 135 degrees or 225 - 315 degrees. Values outside these ranges will not be accepted. See Enhanced Etch Factor Support in the *Allegro PCB SI User Guide* for additional information.

	Diff Coupling Type	Specifies a coupling type. Options are: *NONE EDGE BROADSIDE*
		Note: Broadside coupled traces must have bounding planes for setting the coupling type for a layer or layer pair. Single line trace impedances on each of the two layers should be approximately equal.
	Diff Spacing um	Defines the spacing to use for edge-coupled differential impedance calculations and the trace layer pairing for BROADSIDE coupling. The impedance automatically recalculates to correspond to changes you make in the *Line Width* column. When a different unit of measure is entered in addition to the number, the entered value is converted to the currently selected unit of measure. The default impedance is read from the line-to-line spacing value specified in the active spacing rule set (*Setup – Constraints*)
	Diff Z0 Ohm	Specifies the calculated (or user-defined) differential impedance value for the selected layer. This value is for *what-if* analysis to arrive at a working cross-section, contained within the Cross-section Editor dialog box. The default differential impedance value is calculated using the current line width and spacing, coupling type, and the stack-up geometry.
	SI Ignore	Excludes the layer from Signal Integrity trace model generation and analysis.

Drill Chart Viewer

You can right-click anywhere in the drill chart viewer to display the context-sensitive menu.

Option	Description
Draw Options	Displays *UI Options* dialog box.
Drill Holes Direction Info	Displays *Drill Chart* dialog box that shows drill direction for each drill.

Save Configuration	Allows you to save stackup viewer configuration file (.cnfg).
Load Configuration	Allows you to load stackup viewer configuration file (.cnfg).
Reverse Drill Direction	Select a drill in the viewer and right-click to choose Reverse Drill Direction. This option reverses the direction of Buried/Blind/Microvias. The reversal only impacts the ordering of layers used in the NC Drill Legend Chart.
Drill Directions	Select a drill in the viewer and right-click to choose Drill Directions. Displays *Drill Chart* dialog box that shows drill direction for each drill.

Context-sensitive Worksheet Controls

You can right-click anywhere in the worksheet cells to display the context-sensitive menu.

Option	Description
Add Layers	Displays *Add Layers* dialog box to add multiple layers. You can add: • any type of layer • either above or below the selected layer • prefix to the layer name ⚠ You are not allowed to add more than one dielectric layer above Top/Bottom Conductor layer. The maximum number of layers can be added are 10.
Add Layer Pair Above	Adds a pair of layer to the stackup above the selected layer. ⚠ You are not allowed to add more than one dielectric layer between the outer Surface layer and the Top/Bottom Conductor layer.

Add Layer Pair Below	Adds a pair of layer to the stackup below the selected layer. ⚠ You are not allowed to add more than one dielectric layer between the outer Surface layer and the Top/Bottom Conductor layer.
Add Layer Above	Adds a layer to the stackup above the selected layer. ⚠ You are not allowed to add more than one dielectric layer between the outer Surface layer and the Top/Bottom Conductor layer.
Add Layer Below	Adds a layer to the stackup below the selected layer. ⚠ You are not allowed to add more than one dielectric layer between the outer Surface layer and the Top/Bottom Conductor layer.
Rename	Rename the selected layer.
Remove Layer	Removes the selected layer from the stack. ⚠ You are not allowed to remove the dielectric layer between two conductive layers or any Surface layers.

Related Topics

- xsection

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Single Stackup Support

The primary stackup is the default stackup and represents the largest number of electrical layers (Conductor, plane, and Dielectric). You can turn on or off only non-electrical layers. Areas of the design not represented by a zone name source the Primary stackup.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Multiple Stackup Support

The Cross-section Editor also supports multiple stackups definitions for electrical and non-electrical layers such as Soldermask and Coverlay along with default Primary stackup. The non-electrical layers (mask and coating layers) are used in rigid, flex or rigid-flex applications. The Cross-section Editor provides total thicknesses for each stackup in terms of accumulated electrical layers as well as an option with mask layer thicknesses. You can add non-electrical layers above or below the surface layers (Top and Bottom).

To view the multi-stackup mode, enable *View – Multi Stackups mode.*
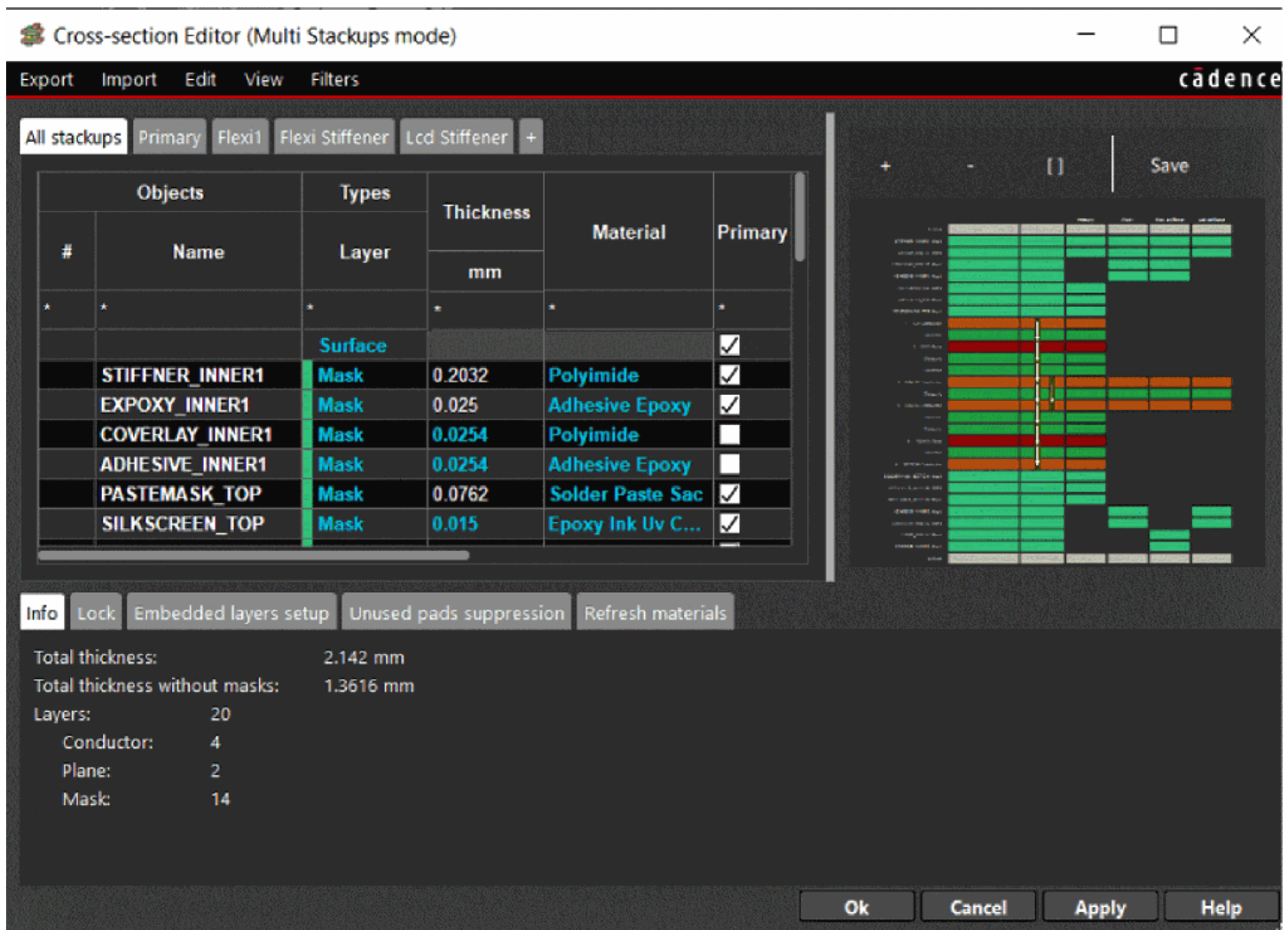
Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Updating the Cross-Section Editor with the Latest Materials Information

Perform the following steps to update the cross-section editor with information on latest materials:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. Click to open *Refresh Materials* tab.
   A menu of refresh options appears.

3. Choose an option from the menu to refresh a desired material property.
   or
   Select *All Values* to update all material properties in the worksheet.

4. Click *Refresh Materials.*
   The `materials.dat` file is read and the worksheet is updated.

> ⚠ *Dielectric Constant* and *Loss Tangent* values update for Dielectric layers only. These properties are never updated for Conductive layers, even if you choose the *Refresh All Values* option from the menu.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Adding Cross-Section Layers

> ⓘ To avoid performance issues when adding layers, you need to first set a sufficient number of planes in the stack (typically every 4th layer).

To add cross-section layers, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.
   This dialog box displays one line for each layer of the layout cross-section. The lines are in the physical order of the layers, from TOP/SURFACE to BOTTOM/BASE as they exist in the layout.

2. In the *Objects Name* column, click a layer to choose it.

3. Right-click and choose *Add Layer Above* or *Add Layer Below* from the pop-up menu.
   The layout editor adds a new dielectric layer above or below the existing layer. You can then change the layer name and type as well as other attributes.

   > ⚠ You can add only one DIELECTRIC layer (typically a conformal coating) between the SURFACE layer (TOP or BOTTOM) and the CONDUCTOR layer (TOP or BOTTOM).

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Removing Cross-Section Layers

To remove cross-section layers, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Objects Name* column, click a layer to choose it.

3. Right-click and choose *Remove Layer* from the pop-up menu.

> ⚠ You cannot remove surface layers or a dielectric layer between two conductive layers.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Material

To change layer material, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Material* column of the appropriate layer, click the triangle beside the *Material* field and choose a material to use from the drop-down menu.

   > ⚠ You can add more materials by editing your own `materials.dat` file. Die layers may exist both above and below the package substrate layers.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Layer Type

To change layer type, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Layer* column of the appropriate layer, in the *Type* field and choose a material type to use.

   ⚠ You cannot change either Surface layer. Top and Bottom layers must be conductive. The *Shield* column applies only to Plane layers and should normally remain checked.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Layer Name

To change layer name, perform the following steps:

1. Choose *Setup – Cross-section*.

   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Objects Name* column of the appropriate layer, enter a name.

   Typically, you name a layer for down-stream manufacturing operations such as masking, artwork, or film generation. You cannot change the name of the TOP or BOTTOM layer.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Layer Thickness

To change thickness of a layer, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Thickness* column of the appropriate layer, enter a number representing thickness for that layer.
   You can enter a different unit of measure in addition to the number. Allegro SI then converts the entered value to the currently selected unit of measure.

> ⓘ Zero thickness layers typically cause problems with impedance calculation and may introduce severe performance problems.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Impedance

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Line Width

To change the line-width of a layer, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Width* column of the appropriate layer, enter a line width value representing the line width for that layer.

> ⚠ The units automatically default to the units set in the *Drawing Parameters* area of the Display Preferences dialog box (*Setup – Preferences*).

You can enter a different unit of measure in addition to the number. Allegro SI then converts the entered value to the currently selected unit of measure.

> ⚠ The impedance recalculates automatically based on the changes you make in the *Width* field.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Dielectric Constant

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Impedance

To change the impedance of a ayer, perform the following steps:

1. Choose *Setup – Cross-section.*
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Impedance* column of the appropriate layer, enter an impedance value for that layer. The line width recalculates automatically based on the changes you make in the *Impedance* field.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Electrical Conductivity

- Editing a Frequency Dependent File

Changing the Dielectric Constant

To change dielectric constant of a layer, perform the following steps:

1. Choose *Setup – Cross-section*.

   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears

2. In the *Dielectric Constant* column for the appropriate layer, enter a number representing the specified constant.
   The impedance recalculates automatically.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Editing a Frequency Dependent File

Changing the Electrical Conductivity

To change electrical conductivity of a layer, perform the following steps:

1. Choose *Setup – Cross-section*.

   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. In the *Conductivity* column of the appropriate layer, enter a number and units.

   ⚠ Entries having a valid unit of measure other than mho/cm are interpolated and converted to mho/cm.

Related Topics

- xsection
- Cross-Section Editor Dialog Box
- Cross-Section Editor Dialog Box Controls
- Single Stackup Support
- Multiple Stackup Support
- Adding Cross-Section Layers
- Removing Cross-Section Layers
- Changing the Material
- Changing the Layer Type
- Changing the Layer Name
- Changing the Layer Thickness
- Changing the Line Width
- Changing the Impedance

Editing a Frequency Dependent File

To edit a frequency dependent file, perform the following steps:

1. Choose *Setup – Cross-section*.
   Alternately, run the command `xsection` from the command line. The Cross-section Editor dialog box appears.

2. Highlight the frequency dependent file you want to edit or view.

3. Click the right mouse button to display the context-sensitive pop-up menu.

4. To edit the selected file, select *Edit Frequency Dependent File*.
   The `.material` text file appears in a text editor.
   –or–

5. To view the selected file as a waveform, select *Display Frequency Dependent File*.
   A waveform (in `.sim` file format) of the electrical characteristics of the frequency dependent file appears in SigWave.

Related Topics

- xsection

- Cross-Section Editor Dialog Box

- Cross-Section Editor Dialog Box Controls

- Single Stackup Support

- Multiple Stackup Support

- Adding Cross-Section Layers

- Removing Cross-Section Layers

- Changing the Material

- Changing the Layer Type

- Changing the Layer Name

- Changing the Layer Thickness

- Changing the Line Width

- Changing the Impedance

- Changing the Dielectric Constant

xsection_chart

The `xsection_chart` command is used to generate a cross-section chart displaying the drill span, stacked vias, embedded component legend, and layer information. You can also create the cross-section information in a table format.

Related Topics

- Adding Cross-Section Charts to the Layout

Cross Section Chart Dialog Box

Access Using

- Menu Path: *Manufacture – Cross Section Chart*

Use the *Cross section Chart and Table* dialog box to specify the parameters for creating chart and table.

Chart Tab

Chart Unit	Specify the unit for displaying chart options. The available units are *Mils*, *Inch*, *Microns*, *Millimeter*, and *Centimeter*.
Maximum Chart Height	Specify the total height of the cross-section chart in *Chart unit*.
Dielectric height scale factor	Controls the dielectric size as displayed in the cross-section chart. By default this value is 1.0. In case you want to make dielectric layer thinner to save space, this value can be set to small decimal number, such as 0.5.
X scale factor	Controls the size of the chart along the x-axis. By default, the value is set to 1. To reduce the width of the chart, you can specify the values less than 1.
Text block	Use this field to specify the size of the text block.
Text block name	Use this field to specify the name of the text block.
Chart Options	Use the options in this section to specify the information to be included in the cross-section chart.
Drill span	Select this to display the drill span — includes pin and the via span.
Stacked vias	Select this option to display the stacked vias.
Backdrill span	Select this to display backdrill layer pairs.

Embedded component legend	Select this to display the embedded components placed on internal PCB layers.
Display Options	Lists the layer information that can be included in the cross-section chart.
Drill label	Select this option to display the via span labels for single vias. This information is displayed only when the *Drill span* option is selected.
Layer name	When selected, displays the layer names, such as TOP, BOTTOM, SIG_1 and so on.
Layer type	Select this to display the layer type for each layer — as specified in the *Cross-section Editor* dialog box.
Layer material name	When selected displays the material used for each PCB layer. This is same as the information displayed in the *Material* column of the *C* dialog box.
Individual layer thickness	Select this to display the layer thickness for each layer — as specified in the *Cross-section Editor* dialog box in *Chart unit.*
Thickness tolerance	Select this to display the thickness tolerance for each layer — as specified in the *Cross-section Editor* dialog box.
Embedded Status	Displays the Embedded status of the layer as specified in the *Cross-section Editor* dialog box. Supported values are: • NOT_EMBEDDED • BODY_UP • BODY_DOWN • PROTRUDING_ALLOWED
Embedded attach method	Displays the method used to attach embedded components to the internal layer. This information is available only for layer with embedded status set to BODY_UP or BODY_DOWN. Supported values are: • INDIRECT_ATTACH • DIRECT_ATTACH

Table Tab

Table Unit	Specify the unit for displaying table options. The available units are *Mils, Inch, Microns, Millimeter,* and *Centimeter*.
Table title	Use this field to specify the name of the table. The default name is STACKUP TABLE.
Text block name	Use this field to specify the name of the text block.
Text block	Use this field to specify the size of the text block.
Height expansion	Controls the table size. Set the value in percentage by which you want to increase the size of table. By default this value is set to 0%.
Table notes	Use this field to add notes at the bottom of the table.
Table Column Options	Use the options in this section to specify the information to be included in the cross-section table.
Layer number	Select this to display the layer type for each layer — as specified in the *Cross-section Editor* dialog box.
Layer name	When selected, displays the layer names, such as TOP, BOTTOM, SIG_1 and so on.
Layer type	Select this to display the layer type for each layer — as specified in the *Cross-section Editor* dialog box.
Layer material name	When selected displays the material used for each PCB layer. This is same as the information displayed in the *Material* column of the *Cross-section Editor* dialog box.
Thickness tolerance	Select this to display the thickness tolerance for each layer — as specified in the *Cross-section Editor* dialog box.

Adding Cross-Section Charts to the Layout

To add a cross-section chart to your design,

1. Choose *Manufacture – Cross Section Chart.*
   Alternately, run `xsection_chart` from the command line. The Cross Section Chart dialog box appears.

2. In the *Chart* tab, enter the values to specify the chart size and select appropriate options to display the required information.

3. Click *OK.*

To add a cross-section table to your design,

1. Run `xsection_chart`.

2. In the *Table* tab, enter the values to specify the table size and select appropriate options to display the required information.

3. Click *OK.*

The chart and table are generated and placed on the left lower corner of the drawing. If you regenerate the chart (or table), it is placed at the last location.

To change the location of the chart (or table), do the following:

- Select the chart (or table) and text as a group, and move it to a desired location.

- Delete the chart (or table) as a group, and add a new chart (or table) to the new location.

Related Topics

- xsection_chart

xsymbol

The `xsymbol` command is used in conjunction with `property edit` to locate objects by symbol name, with `show element` to display information on the named objects, and on certain Edit commands. It differs from the `symbol` command in that the `xsymbol` action is deferred until you run `xname_flush`. This allows you to find/choose multiple symbols.

Related Topics

- Displaying Information for Specified Symbols
- Choosing Objects for Editing using the Xsymbol Command

Xsymbol Dialog Boxes

Depending on the command you run `xsymbol` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Editing using the Xsymbol Command

Displaying Information for Specified Symbols

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose the object type *Symbol* in the *Find* filter.

3. Type `xsymbol` <*symbol name*> at the console window prompt.

4. Repeat step 2 for each additional symbol instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.
   The Show Element display window for the specified symbols appears.

Related Topics

- xsymbol

Choosing Objects for Editing using the Xsymbol Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the object type *Symbol* in the *Find* filter.

3. Type `xsymbol` <*symbol name*> at the console window prompt.

4. Repeat step 2 for each device type which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the symbols for the selected objects.

To choose objects for other Edit commands, run the appropriate supported command; for example, `copy`.

Related Topics

- property edit

- xsymbol

- Xsymbol Dialog Boxes

xsymtype

The `xsymtype` command is used in conjunction with `property edit` to locate objects by symbol type, with `show element` to display information on the named objects, and on certain Edit commands. It differs from the `symtype` command in that the `xsymtype` action is deferred until you run `xname_flush`. This allows you to find/choose multiple symbol types.

Related Topics

- Displaying Information for Specified Symbol Types
- Choosing Objects for Editing using the Xsymtype Command

Xsymtype Dialog Boxes

Depending on the command you run `xsymtype` with, the following dialog boxes are displayed:

- Show Element

- Edit Property

- Show Properties

Related Topics

- Choosing Objects for Editing using the Xsymtype Command

Displaying Information for Specified Symbol Types

Perform the following steps to display information on the named objects:

1. Run the `show element` command.

2. Choose the object type *Symtype* in the *Find* filter.

3. Type `xsymtype` *<symbol type name>* at the console window prompt.

4. Repeat step 2 for each additional symbol instance on which you want information.

5. When done, type `xname_flush` at the console window prompt.
   The Show Element display window for the specified symbols appears.

Related Topics

- xsymtype

Choosing Objects for Editing using the Xsymtype Command

Perform the following steps to choose the objects whose properties are to be edited:

1. Run the `property edit` command.

2. Choose the object type *Symtype* in the *Find* filter.

3. Type `xsymtype` *<symbol typename>* at the console window prompt.

4. Repeat step 2 for each device type which you want to edit.

5. When done, type `xname_flush` at the console window prompt.
   The Edit Property and Show Properties dialog boxes are displayed.

6. Edit the symbols for the selected objects as described in property edit.

To choose objects for other Edit commands, run the appropriate supported command; for example, `copy`.

Related Topics

- xsymtype
- Xsymtype Dialog Boxes

xymode

An internal Cadence engineering command.