cadence®

W Commands

Product Version 23.1
September 2023

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

1 8
W Commands 8
waive drc 9
Waive DRC Command: Options Panel 10
Waiving a DRC Error 11
wbedit 14
Assign Nets Dialog Box 16
wbedit Command: Options Panels 17
Wire Bond Edit mode - Perform Auto Bonding Command: Options Panel 18
Wire Bond Edit mode - Populate with Fingers Command: Options Panel 20
Wire Bond Edit mode - Add Non-Standard Wire Bond command: Options Panel 21
Wire Bond Edit mode - Change Characteristics Command: Options Panel 22
Wire Bond Edit mode - Create Ring Command: Options Panel 25
Wire Bond Edit mode - Edit Parameters for Power/Ground Rings: Options Panel 26
Wire Bond Edit mode - Add Flag: Options Panel 27
Wire Bond Edit mode - Edit Routing Stubs: Options Panel 29
Wire Bond Edit mode - Add Jumper Command: Options Panel 30
Generic Wire Bond Design Tasks 31
Element Selection in Find Filter and Corresponding Wire Bond Tasks 33
Performing Auto Wire Bonding 36
Adding a Bond Finger Guide Path 38
Adding a Die Flag 39
Editing or Copying the Bond Finger Guide Path 41
Moving the Bond Finger Guide Path 42
Creating the Best Fit Guide Path 43
Adding Bond Fingers and Routing Stubs 44
Editing Routing Stubs 46
Adding Non-Wired Bond Fingers by Selecting a Bond Finger 47
Adding Non-Wired Bond Fingers by Selecting a Die Pad 48
Adding Non-Wired Bond Fingers by Selecting a Guide Path 49
Adding Non-Standard Wire Bonds 50
Connecting a Die to Non-wired Bond Fingers 51
Attaching Multiple Wire Bonds to One Bond Finger 52

Defining Padstacks for Bond Fingers 54
WireBonding from a Die to an Interposer 56
WireBonding from an Interposer to the Substrate 57
Deleting Bond Fingers 59
Moving Bond Fingers 60
Swapping Bond Fingers 62
Spacing Bond Fingers Evenly 63
Adjusting Bond Fingers to Meet Minimum DRC Requirements 64
Centering Bond Wires Between Bond Fingers 65
Adding or Deleting Routing Channels 66
Creating a Merged Finger Shape 67
Removing a Merged Finger Shape 68
Creating a Ring Segment from a Set of Bond Fingers 69
Changing the Characteristics of Wire Bonds 70
Redistributing Bond Fingers Across Multiple Bond Finger Guide Paths 71
Adding a Jumper between Bond Fingers 72
Populating a Guide Path with Bond Fingers 73
Deleting Bond Wires 75
Moving, Centering, or Swapping Bond Wires 76
Adjusting Bond Wires to Meet Minimum DRC Requirements 76
Splitting or Reconnecting Power/Ground Ring 78
Assigning Nets to a Ring or Ring Segment 79
Editing Ring Parameters for a Power/Ground Ring 80
window center 81
Selecting the Location in the Center of the Window 82
window fit 83
window in 84
window out 85
window points 86
Specifying an Area of a Design 87
window world 88
wirebond add 89
Wirebond Add Command: Options Panel 90
Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers) 95
Bond Finger Padstack Dialog Box 97
Wirebond Add Procedures 99

wirebond add nonstandard 100
Wirebond Add Nonstandard Command: Options Panel 101
wirebond edit 103
Wirebond Edit Command - Move Command: Options Panel 104
Add Routing Channel Dialog Box 105
Wirebond Edit Procedures 106
wire bond escape 107
Wirebond Escape Command: Options Panel 108
Wire Bond Escape: Main Pop-up Menu 111
Die Escape: Escape Direction Dialog Box 113
Escaping from a Wire Bond Die 115
wirebond export 117
Wire Bond Export Dialog Box 118
Exporting Parameters from a Design 120
XML-Based Settings File 121
wirebond import 131
Wire Bond Import Dialog Box 132
Importing Setting from a Pre-Release 16.3 XML File 134
Importing a Wire Bond Pattern from Another Design 135
wirebond lock 136
Running the Wirebond Lock Command 137
wirebond manage guide paths 138
Wirebond Manage Guide Paths - Add Guide Command: Options Panel 139
Wirebond Manage Guide Paths - Redistribute Fingers Command: Options Panel 141
Wirebond Manage Guide Paths Procedures 142
wirebond purge groups 143
Purge Unused Wire Bond Groups Dialog Box 144
Purging Unused Wire Bond Groups from your Design 145
wirebond reports 146
Wire Bond Reporting Dialog Box 147
Generating a Wire Bond Report 151
wirebond select 152
Dialog Boxes and Options Panels for the Wirebond Select Command 153
Advanced Selection Filtering Dialog Box 154
Advanced Selection Filtering 155
Wirebond Select Command - Perform Auto Bonding Command: Options Panel 156
Wirebond Select Command - Populate with Fingers Command: Options Panel 158

Wirebond Select Command - Change Characteristics Command: Options Panel 159
Wirebond Select Command - Create Ring Command: Options Panel 162
Wirebond Select Command - Edit Routing Stubs: Options Panel 163
Wirebond Select Command - Add Jumper Command - Options Panel 165
wirebond settings 166
Wire Bond Settings Dialog Box 168
Bond Finger Labels Dialog Box 174
Global Wire Bond Constraints Dialog Box 176
Pre-defined Wirebond Settings Dialog Box 180
Wire Bond Profile Editor 182
Setting Wire Bond Parameters 186
Adding a Wire Profile 187
Initializing Your Wire Profiles 189
Copying a Wire Profile 190
Deleting a Wire Profile 191
Editing a Wire Profile 192
Refreshing a Wire Profile's Definition from the Master Definitions File 193
Purging Unused Wire Profiles from a Design 194
wirebond soldermask create 195
Wirebond Soldermask Create Command: Options Panel 196
Creating Soldermask Shapes for Bondpads Automatically 198
wirebond tack point 199
Wirebond Tack Point Command: Options Panel 200
Running the Wirebond Tack Point Command 202
Automatically Spreading All Wires on Fingers 203
Moving the Tack Point Along the Pad Axis 204
Moving the Tack Point Freely Within the Pad 205
Copying an Existing Tack Point Offset 206
Removing a Tack Point Offset 207
wirebond unlock 208
Running the Wirebond Unlock Command 209
wirebond uprev 210
Running the Wirebond Uprev Command 211
wire bond via estimation 212
Wire Bond Via Estimation Dialog Box 213
Set Via-Pad Size Dialog Box 217
Running the Wirebond Via Estimation Command 219

workflow 220
Workflow Manager 222
Importing a Partition into a Master Database 226
Exporting a Partition from a Master Database 227
Refreshing a Partition 228
Retracting a Partition 229
Previewing Partitions Prior to Export from The Master Database 230
Appending Notes to a Master or Partition Database 231
Reporting Connection Statistics 232
Recovering an Obsolete Partition 233
world cancel 234
world done 235
world drawing2port 236
world find next 237
world find previous 238
world move 239
Changing the Position of the Current Window over the Design 240
world oops 241
world resize 242
Zooming into Current Window over the Design 243
write 244
Saving the layout to a file with a different name 245

1

W Commands

waive drc	wbedit	window center
window fit	window in	window out
window points	window world	wirebond add
wirebond add nonstandard	wirebond edit	wire bond escape
wirebond export	wirebond import	wirebond lock
wirebond manage guide paths	wirebond purge groups	wirebond reports
wirebond select	wirebond settings	wirebond soldermask create
wirebond tack point	wirebond unlock	wirebond uprev
wire bond via estimation	workflow	world cancel
world done	world drawing2port	world find next
world find previous	world move	world oops
world resize	write	

waive drc

The `waive drc` command sets aside design rule violations to meet design requirements. You can flag these violations as acceptable, and attach an explanatory comment to ensure that those working with the design later on understand the rationale behind their existence. When you waive a DRC error, and you enable the visibility of DRC error markers in the *Color and Visibility* dialog box, the markers change their appearance to easily distinguish waived DRC errors from active DRC errors. A report lists all waived DRC errors, available by choosing *Tools – Reports – Waived Design Rules Check Report*.

The command is also an option on the right-mouse button pop-up menu, available in an active application mode, which provides an intuitive environment in which commands used frequently are readily accessible, based on a selection set of design elements you have chosen. This command functions in a pre-selection use model, in which you choose an element first, then right click and execute the command. The valid object is DRC Errors.

⚠ If you window select a group of DRC errors that contain both waived and restored DRCs, both *Waive DRC* and *Restore DRC* appear in the pop-up menu, and you can use either as required. To restore all the waived DRCs within the group, right click and choose *Quick Utilities – Restore All Waived DRCs* from the pop-up menu.

Related Topics

- show waived drcs
- blank waived drcs
- restore waived drc
- Creating Design Rules
- Waiving a DRC Error

Waive DRC Command: Options Panel

Access Using

- Menu path: *Display – Waive DRCs – Waive*
- Toolbar Icon

The displayed information duplicates that which appears in the *Show Element* dialog box but does not appear when you are working in an application mode.

Constraint Name	Displays the name of the design rule that produced the violation.
Constraint Set Name	Displays the name of the constraint set in which the DRC error exists.
Constraint Type	Specifies the type of design rule.
Subclass	Specifies the layer on which the DRC error exists.
Origin	Specifies the X, Y coordinates of the DRC error.
Constraint Value	Specifies the value as defined in the constraint rule.
Actual	Specifies the actual value of the offending element on the board.
Default comment	Adds a comment to a waived DRC error. The comment appears in the information in the *Show Element* dialog box.

Waiving a DRC Error

To waive a DRC error, do the following:

1. Hover your cursor over a DRC error marker or window select a group of DRCs. The tool highlights it, and a data tip identifies its name.
2. Right click and choose *Waive DRC* from the pop-up menu.
   The *Comment for Waived DRCs* dialog box displays, where you can add a comment about the waived DRC error (optional).
3. Enter a comment if desired, and click OK.
   The DRC error marker rotates 90 degrees and assumes the color set in the *Color and Visibility* dialog box. The waived DRC marker is visible only if you enable both of the following:
   - the *Waived DRCs* check box in the *Display* tab of the *Design Parameter Editor*, available by choosing *Setup – Design Parameters* (`prmed` command)
   - DRC visibility in the *Color and Visibility* dialog box

   ⚠ The DRC error count available in the *Status* tab of the *Status* dialog box automatically updates.

Example

Correct by design, nets DGND, AGND, GND connect at the same X,Y location producing a DRC error. Waiving a DRC error lets you flag the error as acceptable in the design as shown.

DRC Error Before Waiving

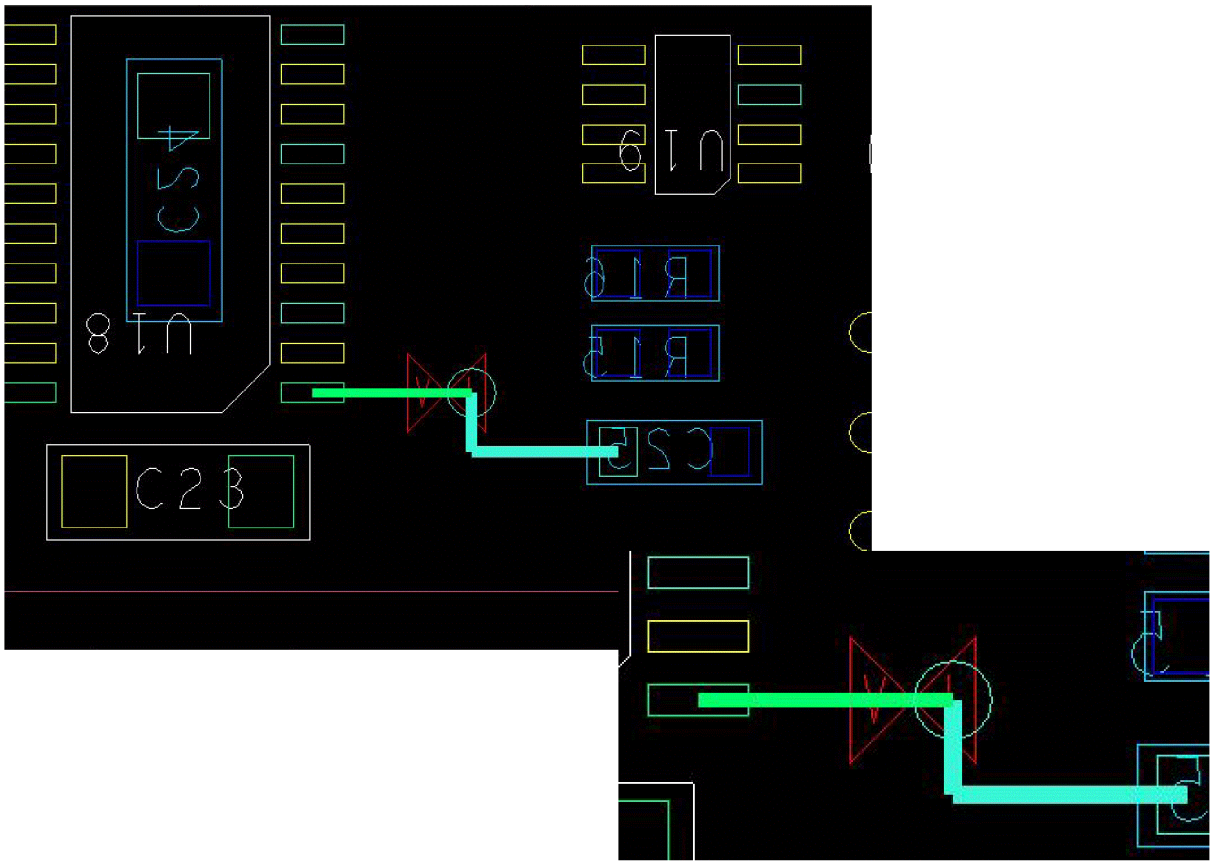
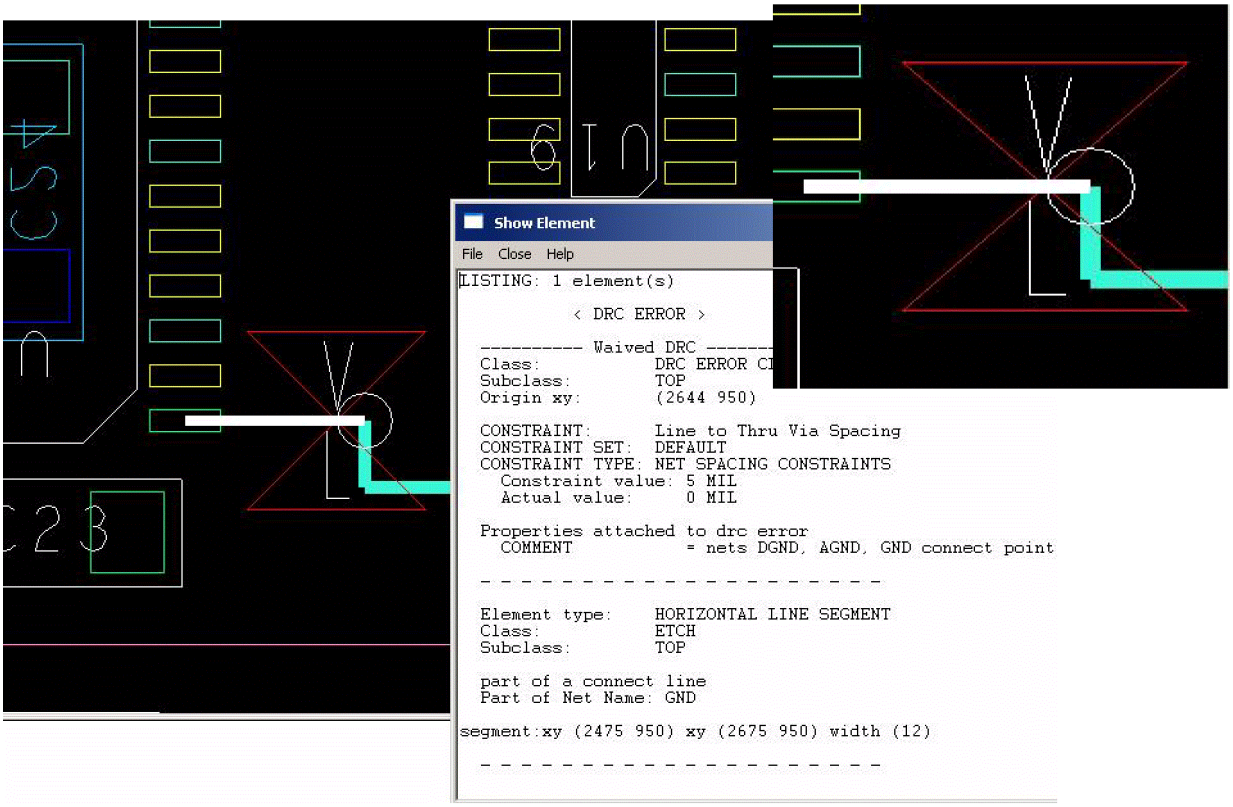


Figure 1.1: DRC Error After Waiving



Related Topics

- waive drc
- prmed

wbedit

The `wbedit` command enables Wire Bond Edit application mode that customizes your environment to perform wire bond editing tasks, such as adding wire bonds to die pins, moving and swapping bond fingers and wires, and editing and moving guide paths. An application mode provides an intuitive environment in which commands used frequently in a particular task domain, such as editing or moving, are readily accessible from right-mouse- button popup menus, based on a selection set of design elements you have chosen.

In conjunction with an active application mode, your tool defaults to a pre-selection use model, which lets you choose a design element (noun), and then a command (verb) from the right-mouse-button popup menu. This pre-selection use model lets you easily access commands based on the design elements you've chosen in the design canvas, which the tool highlights and uses as a selection set, thereby eliminating extraneous mouse clicks and allowing you to remain focused on the design canvas.

In addition, right-clicking a single item automatically selects that item and gives the appropriate context-sensitive menu. As a result, there is no need to click to select it first.

You can also configure the default drag and double-click operations in the user preferences dialog. For example, you can select the move operation as the default when a bond finger is dragged. If this is configured, you can click on a finger and, while the button is depressed, drag to the new location and release without using the context-sensitive menu.

Use *Setup – Application Mode – None* (`noappmode` command) to exit from the current application mode and return to a menu-driven editing mode, or verb-noun use model, in which you choose a command, then the design element.

Access Using

- Menu path: *Setup – Application Mode – Wire Bond Edit*
- Toolbar icon: 

Related Topics

- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- Advanced Selection Filtering Dialog Box

Assign Nets Dialog Box

This dialog box appears when you select a ring and choose *Assign Net* from the mouse pop-up menu.

Master net	Specifies the primary net on which to connect this shape. This is the only net allowed to permanently connect to this shape. You can select this in the design using the Find Filter or browsing by net name. If you choose a master net that does not have a voltage property, the tool prompts you to enter a voltage.
Shorted nets	Specifies the list of all the nets shorting to this ring legally. Common causes include multiple nets on the die mapping to a single net in the package or a temporary setting that allows the creating of split power and ground rings. ⚠ To ensure against illegal shorts, net shorts established here apply for the wire bond tool only. To specify a short as legal permanently, you must use the *Route – Create Net* or *Route – Define Net Short* commands
Add	Browse for, and add, another net to the list of allowed shorted nets.
Remove	Remove the selected shorted nets from the list.
OK	Accepts changes to the database and dismisses the dialog box.
Cancel	Cancels the changes and dismisses the dialog box.
Help	Displays help for this command.

Related Topics

- wbedit
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

wbedit Command: Options Panels

To modify options for wire bond edit mode commands you may choose one of the following:

Wire Bond Edit mode - Perform Auto Bonding Command: Options Panel	Wire Bond Edit mode - Populate with Fingers Command: Options Panel	Wire Bond Edit mode - Add Non-Standard Wire Bond command: Options Panel
Wire Bond Edit mode - Change Characteristics Command: Options Panel	Wire Bond Edit mode - Create Ring Command: Options Panel	Wire Bond Edit mode - Edit Parameters for Power/Ground Rings: Options Panel
Wire Bond Edit mode - Add Flag: Options Panel	Wire Bond Edit mode - Edit Routing Stubs: Options Panel	Wire Bond Edit mode - Add Jumper Command: Options Panel

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Perform Auto Bonding Command: Options Panel

These settings appear in the *Options* panel when you run the command to perform auto bonding. See Generic Wire Bond Design Tasks.

Bond wires	
Profile	Specifies the wire bond profile to use for auto bonding. By default, this field is set to the profile specified as default in wire bond settings.
Power/Ground rings	
Create rings	Specifies whether the tool creates power and ground rings. The rings are placed in relation to the die in order of their increasing voltage; the ring with the lowest voltage being closest to the die and the ring with the highest voltage being farthest from the die. If two rings have the same voltage, they are sorted in the alphabetical order. The default setting is unchecked. ⚠ The tool creates rings with rounded corners, using the default setting of the `pring wizard` command.
Dist. to 1st ring	Specifies the distance between the die edge and the first ring boundary. By default, this is 7.5% of the minimum of die width and die height (same as in the `pring wizard` command).
Ring width	Specifies the width of the ring.
Dist. between rings	Specifies the distance between the two rings. By default, this is 7.5% of the minimum of die width and die height (same as in the `pring wizard` command).
Guide paths	
Linear path	Specifies whether the tool creates a linear guide path. If you do not check this box, the bond finger guide path is an arc. By default, this box is not checked.
Row bond	Check this box to enable row bonding. The tool calculates the pin rows on each die side and creates the same number of guide paths as the number of pin rows per side. It does not consider the value you enter for the number of guide paths in the *Options* panel. The die or diestack is auto-bonded with the row closest to the die edge being bonded to the closest guide path to the die. Pin rows are determined by pin (x,y) location. If you enable row bonding, the *Paths per die side* field is disabled. By default, row bonding is *Off*.

Paths per die side	Specifies the number of paths that the tool creates per die side.
Dist. to 1st path	Specifies the distance between the first guide path on a side to the die edge if you do not create rings, or to the outer ring boundary if you create rings. By default, this setting equals the value of the *Dist. to 1st ring* field.
Dist. between paths	Specifies the distance between guide paths on one side. By default, this setting equals the value of the *Dist. between rings* field.
Auto bond	Starts the auto bond process. ⚠ The tool uses the default bond finger padstack (`wirebond settings` command) for the bond fingers.
Done auto bond	Completes the process and exits the command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Populate with Fingers Command: Options Panel

Spacing	Specifies the bond finger-to-bond finger spacing that the tool should use. This value defaults to the general bond finger-to-bond finger spacing value in the DRC constraints. You can override this value to account for same net bond fingers, routing channels, and so on. If you enter a value less than the DRC constraint value, DRCs result.
Bond Finger	Specifies the bond finger padstack that the tool uses for adding fingers to the selected paths. Choose from the drop-down list or click *Add* to create a new padstack. This value defaults to the default finger padstack specified in the wire bond settings.
Wire Profile	Specifies the wire profile that you want to use when you add bond wires to the bond fingers you create with this command.
Create bond fingers	Click this button to save the bond fingers to the design.
Number of bond fingers that will be created given current settings	A status indicator that indicates the total number of fingers created, using the current settings.
Number of new DRCs based on current settings	A status indicator showing the number of new DRCs resulting from the bond finger creation, using these settings.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Add Non-Standard Wire Bond command: Options Panel

Snap to pad origin	If you check this box, when you select a padstack-based object as either the start or end of the wire, the tool snaps the wire end to the pad origin location. If you do not enable this option, the tool connects to the location on the pad where you click. The default setting is On.
Snap to shape center line	Snaps the selected pick to the center line of the shape the wire is connecting to. This is not selected by default.
Profile	Specifies the wire profile to use. You can select a profile from the drop-down list or enter a new name. This setting defaults to the default profile specified in the wirebond settings command or Nonstandard if there are no available profiles.
Padstack for cline connections	Specify the padstack to use when connecting a wire directly to a cline end.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Change Characteristics Command: Options Panel

You can change characteristics for a set of bond fingers or change a wire profile using the *Options* panel. You can see the resulting changes in the bond pattern in the Design Window as you make the changes.

Pattern	
Style	Specifies the wire bond pattern. Choices are: • *Orthogonal – Specifies a pattern of* 0-degree bond wire angles and bond pads. • *Equal Wire Length* – Specifies that wire bond angles are variable, but all wire bond lengths are equal. • *On-Path* – Specifies that all wire bonds stay on the bond finger guide path. • *Free Placement* – Does not snap to any path. > With this option, the fingers are never automatically pushed or shoved due to bubbling as they have no associated path of movement. This field determines the placement rules applied based on the path that these elements follow. The default setting is *On-Path unless there are no guide paths; then the default setting is Equal Wire Length*. This field is read-only if you select *Bond wires* in the Find Filter.
Length	Specifies the wire length. This field is available only if you select *Equal Wire Length* as a style. ⚠ For non-wired bond fingers, this length is approximated for the placement of non-wired bond fingers since there is no measurable length of wire from pin to bond finger. For non-wired bond fingers that have the *On Path* pattern but have no paths because they were removed after the group was created, the placement is approximate and not based on the value in the *Length* field since there is no path to snap to.
Wire	

Profile	Specifies the wire bond profile (including diameter) to associate with this wire. A profile determines the approximate 3D path that the wire bond follows. To specify the name of a new profile, enter the name in this field and then use the *Route – Wirebond – Settings – View/Edit wire profiles* to create the profile in the profile editor. Then you can select the profile from the pull-down menu. This field is read-only if no wire has been selected.
Finger	
Pad	Lists the available padstacks that you select for the bond fingers. The default padstack is specified in the Wire Bond Settings dialog box (see the wirebond settings command). This field is read-only if no finger has been selected.
Add	Displays the Finger Padstack dialog box which allows you to define a new finger padstack.
Align	Specifies the orientation to use when placing bond fingers. Choices are: • *Aligned with Wire* • *Orthogonal to Die Side* • *Orthogonal to Guide* • *Pivoting Ortho to Guide* • *Average Wire Angle* • *Constant Angle* • *Match CW Neighbor* • *Match CCW Neighbor* The default setting is *Aligned with Wire*. Choose the *Pivoting Ortho to Guide* option for oblong or rectangular bond fingers. It lets you avoid the situation when the wire is almost perpendicular to the die side and the bond wire crosses the boundary of the bond finger over the rounded end of the bond finger. As the bond wire angle increases, the bond wire starts to cross the bond finger boundary on one of its straight edges. With this option, the tool pivots the bond finger slightly so that the bond wire crosses over the rounded end. This field is read-only if no finger has been selected.
Angle	Specifies the value for the angle alignment. This field is read-only unless you select *Constant Angle* alignment.

Snap	Controls which point on the bond finger snaps to the guide path. Because bond fingers vary in size, sometimes it is necessary to control how they follow the path so that bond wire lengths are equal and you achieve the smoothest possible pattern. Options are: • *Center of Finger* • *Finger Origin* • *Near End* (the tip of the bond finger closest to the die edge) • *Far End* (the tip of the bond finger farthest from the die edge) • *Nearest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is closest to the die) • *Farthest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is farthest from the die) The default setting is *Finger Origin*. This field is read-only if you have not selected a bond finger.
Apply changes and exit	Applies the specified changes to the selected fingers or bond wires and exits the change characteristics mode.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Create Ring Command: Options Panel

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

The *Options* panel displays the following parameters when you use the `wirebond select` (or `wirebond edit`) command with the *Advanced – Create Ring* menu item to create a ring segment from a set of placed bond fingers.

Clearances	
Inside	Specifies the expansion distance to apply to the inside edge of the shape (end of the fingers pointed towards the die edge). The default value is 0 UM.
Outside	Specifies the expansion distance to apply to the outside boundary of the shape (end of the fingers pointed away from the die). The default value is 0 UM.
Ends	Specifies the expansion distance to apply to the two ends of the ring segment when generated. The default value is 0 UM.
Create rectangular shape	Click this box to generate a plain rectangle based on the extents of the convex hull computed. Using this option makes it easier to slot ring segments together, modify, and get a clean pattern. The default value is disabled.
Generate	Creates the new ring section, its net assignment (same as the old bond fingers), and the replacement of old fingers with shape tack connections.

Wire Bond Edit mode - Edit Parameters for Power/Ground Rings: Options Panel

Ring distance from die/stack edge	Specify the distance from the die or stack edge to the ring.
Ring width	Specify the width of the ring.
Update ring soldermask shape	Select to update the soldermask shape for the ring.
Update ring	Click to update the selected ring with the specified settings.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Add Flag: Options Panel

Layer	Specify the layer on which you want to add the flag. The default is Top_Cond.
Net	Specify the net on which you want to add the flag. Specify *<No Net>*, the default value, for a dummy flag.
Group flag with die symbol	Check to group the flag with the die symbol. Not selected by default.
Solder opening	Specify the solder opening, which is the distance from the edge of the flag to the edge of the opening. Positive value means the solder mask is larger than the flag, negative means the solder mask is smaller. Not selected by default.
Style	Specify the style of the die flag. You can select either *Solid* or *Wheel*. The default value is *Solid*.
Edge dist	Specify the distance from the edge of the die. By default, the value is `0.00 UM`.
Round corners	Check if you want a flag with rounded corner. Selected by default.
Ring width	Specify the width of the flag ring. Default is `100.00 UM`. Available only when wheel is selected in Style.
Center area	Enabled only if *Wheel* is selected as Style. Available only when wheel is selected in Style.
Shape	Specify the shape of the center area of the flag. Default is *Circle*.
Width	Specify the width of the center area. Default is `500.00 UM`.
Height	Specify the height of the center area. Default is `500.00 UM`.
Spokes	Enabled only if *Wheel* is selected as Style.
Width	Specify the width of each spoke. Default is `100.00 UM`.
Count	Specify the number of spokes. Default is `8`.
Create Flag	Click to create the specified flag.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Edit Routing Stubs: Options Panel

Routing Stubs	
Outer	Check this box to modify or add a stub from the bond finger in the direction away from the die edge. Uncheck the box to remove stubs in this direction. The default setting is *Off*, which causes the *Length* and *Padstack* fields to be disabled.
Inner	Check this box to modify or add a stub from the bond finger in the direction toward the die edge. Uncheck the box to remove stubs in this direction. The default setting is *Off*, which causes the *Length* and *Padstack* fields to be disabled.
Outer / Inner Length	Specifies the length of the stub, which is measured from the end of the bond finger from which the stub exits. By default, it is twice the width and length of the default bond finger padstack to ensure that there is clearance from the bond finger. The tool always aligns the stubs with the bond finger's rotation to ensure a good routing from the die for routing and manufacturing. The tool always aligns the stubs with the bond finger's rotation to ensure optimal routing from the die for manufacturing and wire bond assembly. If you want an off-angle fanout, you can change it using the *Edit – Vertex* command. If you have multiple length stubs for the selected items, ** is displayed in the *Outer* or *Inner* fields.
Outer /Inner Via Pad	Specifies the padstack to use for the via at the end of the routing stub, if desired. By default, this field is set to <*No Via*>, and no via is added. If you have multiple via pads for the selected items, ** is displayed in the *Via* fields.
DRC Count	Displays the number of additional DRC errors resulting from the current settings.
Update Routing Stubs	Click this button when you have finished changes to the *Options* panel. This commits the changes to the database and returns you to the idle wire bonder state.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wire Bond Edit mode - Add Jumper Command: Options Panel

Wire	
Allowed multiple wires to finger	Check this box to allow the tool to add multiple bond wires to a bond finger.
Profile	Lets you select the profile for the jumper.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Generic Wire Bond Design Tasks

The tasks for designing a package containing a wire bond die can vary significantly from user to user. However, you must perform certain steps to arrive at a final, optimal wire bond pattern.

You perform most of the following tasks over the course of designing a package. The following section presents a typical order of steps for implementing a single-chip, top-mounted wire bond package.

1. Set wire bond technology constraints. Also configure labels to update automatically when you set up wire bond settings.
2. Configure the wire profiles. See the *Wire Bonding Tool Set* in the *Allegro User Guide: Routing the Design.*
3. Generate power and ground ring s; generate the die flag.
4. Add the *NO_WIREBOND* property to any pins or nets that you do not wanted wire bonded in this design.
   This property tells the wire bond tool to ignore these objects whenever your window selection includes them. By setting this property on your no connect nets and pins, you can automatically exclude them instead of manually creating a temp group and deselecting these items when adding wire bonds to your design.
5. Wire bond power and ground die pins to rings.

   ⚠ When performing wire bond operations such as addition and movements, power and ground ring bonds are placed at minimum clearance to one adjacent wire bond. If you enable the WIREBOND_CENTER_RING_BONDS environment variable (`enved` command), the tool runs a post-process step to attempt to center the power and ground wires between the adjacent wires on both sides. This improves manufacturability of the design, but may cause a slight performance decrease during interactive wire bonding.

6. Split rings based on resulting power and ground wire bonds.
7. Generate initial bond finger guide paths.
   Attempt to wire bond to one guide path or the minimum planned guide paths.
8. Split into multiple paths based on density and cost analysis, if needed.
9. Move outer bond fingers to allow for fan-out of inner guide paths.
10. Adjust spacing for rule-based wire bond crosses.
11. View the wire bond pattern in 3D space with DRCs to scan for assembly or manufacturing problems.
12. Adjust placement to satisfy 3D spacing requirements for wire bonds.

13. Merge the same net fingers into a multi-point bond finger.
14. Adjust placement to meet signal integrity constraints.
15. Place fiducials.
16. Adjust for final manufacturing and assembly rules by checking these rules and manipulating the pattern.
17. Generate labels, reports, diagrams, and other documentation.
18. Generate solder masks and the bonding tool control file.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Wire Bond Use Models

Element Selection in Find Filter and Corresponding Wire Bond Tasks

The following table lists the elements selected in the Find Filter and the corresponding tasks that you can perform in the Wire Bond Edit application mode.

When You Select This Element in the Find Filter...	You Can Perform this Task...
Comps	• Generic Wire Bond Design Tasks • Adding a Bond Finger Guide Path • Adding a Die Flag • Adding Non-Standard Wire Bonds
Pins	• Adding Bond Fingers and Routing Stubs • Adding Non-Wired Bond Fingers by Selecting a Die Pad • Attaching Multiple Wire Bonds to One Bond Finger • Defining Padstacks for Bond Fingers • WireBonding from a Die to an Interposer • WireBonding from an Interposer to the Substrate

Fingers	• Creating the Best Fit Guide Path • Adding Non-Wired Bond Fingers by Selecting a Bond Finger • Editing Routing Stubs • Connecting a Die to Non-wired Bond Fingers • Attaching Multiple Wire Bonds to One Bond Finger • Defining Padstacks for Bond Fingers • Deleting Bond Fingers • Moving Bond Fingers • Swapping Bond Fingers • Spacing Bond Fingers Evenly • Adjusting Bond Fingers to Meet Minimum DRC Requirements • Centering Bond Wires Between Bond Fingers • Adding or Deleting Routing Channels • Creating a Merged Finger Shape • Removing a Merged Finger Shape • Creating a Ring Segment from a Set of Bond Fingers • Changing the Characteristics of Wire Bonds • Adding a Jumper between Bond Fingers

Bond Wires	• Changing the Characteristics of Wire Bonds • Deleting Bond Wires • Moving, Centering, or Swapping Bond Wires • Adding Non-Wired Bond Fingers by Selecting a Die Pad • WireBonding from a Die to an Interposer • WireBonding from an Interposer to the Substrate • Splitting or Reconnecting Power/Ground Ring • Adjusting Bond Wires to Meet Minimum DRC Requirements
Guide Paths (Lines)	• Adding a Bond Finger Guide Path • Editing or Copying the Bond Finger Guide Path • Moving the Bond Finger Guide Path • Creating the Best Fit Guide Path • Adding Non-Wired Bond Fingers by Selecting a Guide Path • Redistributing Bond Fingers Across Multiple Bond Finger Guide Paths • Populating a Guide Path with Bond Fingers
Shapes	• Splitting or Reconnecting Power/Ground Ring • Assigning Nets to a Ring or Ring Segment • Editing Ring Parameters for a Power/Ground Ring

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels

Performing Auto Wire Bonding

You can use this command in the early stage of the wire bonding process to develop a quick prototype of the general finger pattern style for the pin layout. Prior to running this command, be sure to complete settings and constraint parameters (wirebond settings command).

Perform this procedure as many times until you obtain an acceptable pattern.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Be sure that you check *Comps* in the Find Filter.
3. Select a die for bonding.
   In APD, you can select only one die at a time. In the SiP tools, you can select multiple dies belonging to the same die stack.
4. Right-click and choose *Auto Wire Bond* from the pop-up menu.
   The *Options* panel in the Control Panel displays the appropriate parameters. For information about these fields, see wbedit Command: Options Panels.
5. Complete the parameters, based on your design, and then click *Auto bond.*
   Paths and shapes are dynamically updated and shown in the Design window as you change the settings.
   The tool wire bonds the specified die, based on bond finger guide path and ring information as well as the constraints you set. For multiple paths on one die side, the tool distributes the fingers on all guide paths on each die side, starting from the inner path to the outer paths, and from the center reference pin.
   If you enabled row bonding, the *Paths per die side* field is disabled.
   Power and ground pins are bonded to power and ground rings if they exist in the design.

⚠ Parameters and all objects created during the operation of the command (such as bond finger, bond wire, guide path, and ring information) are logged to a file.

1. Click *Done auto bond* to complete the process and exit the command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding a Bond Finger Guide Path

You need to add a bond finger guide path when you select the wire bond pattern styles of *Orthogonal* or *On Path*. You can also perform this task using the `wirebond manage guide paths` command.

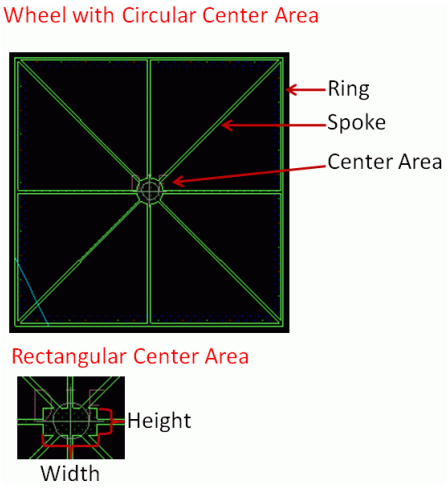
1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Be sure that you check *Comps* in the Find Filter.
3. Select a die or discrete component.
4. Right-click and choose *Add Guide Path* from the pop-up menu.
   The *Options* panel in the Control Panel displays the appropriate parameters. The tool draws a line in the Design Window that uses the current settings.
5. Modify the settings if needed, and then click *Create Guide Path* to make the line permanent.
   With this command, you can create basic paths easily and then edit and refine them as the wire bond pattern evolves.
6. Continue to perform step 5 until you are done; then click *Done adding guides.*

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- Wirebond Manage Guide Paths - Add Guide Command: Options Panel
- wirebond manage guide paths

Adding a Die Flag

You can either add a solid flag or a wheel flag with spokes and a central area that can be of different shapes: circle, octagon, rectangle, or square. For a wheel, you can specify different settings for the shape, width of the rings, and width and height of the central area as shown in the following figure.



1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Be sure that you check *Comps* in the Find Filter.
3. Select the dies you want to add flags.
4. Choose *Add Flag* from the pop-up menu.
   You can also enter the `wirebond add flag` command.
5. Modify the parameters in the *Options* panel according to the task you are performing.
6. Click *Create Flag*.
   You can undo the flag and recreate it if you want (`Ctrl+Z`).

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Editing or Copying the Bond Finger Guide Path

You can also perform these tasks using the wirebond manage guide paths command.

To edit or copy a bond finger guide path:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Lines* in the Find Filter.
3. Select the bond finger guide path.
4. Choose *Edit* or *Copy* from the pop-up menu.
5. Using the cursor, either move the vertices or change the arc radius of the segment until you are satisfied, or move the cursor until you find the location for the copied bond finger guide path, and then click in the Design Window
   -or-
   Use the `ix/iy` command and enter values in the console window.
6. choose *Done* from the pop-up menu. Otherwise, continue to perform other tasks within this command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Moving the Bond Finger Guide Path

You can also perform this task using the wirebond manage guide paths command.

To move a bond finger guide path:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Lines* in the Find Filter.
3. Select the bond finger guide path.
4. Choose *Move* from the pop-up menu.
5. Move the cursor until you find the location for the copied bond finger guide path and then click in the Design Window
   -or-
   Use the `ix/iy` command and enter values in the console window.
6. Choose Done from the pop-up menu. Otherwise, continue to perform other tasks within this command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Creating the Best Fit Guide Path

If you have a design with bond fingers (*Free Placement* style), and you want to snap them to a guide path, or if a guide path was accidentally removed from the design, you may want the tool to recompute a guide path for the affected fingers. You can tell if a guide path has been accidentally removed from the design because when you are performing wire bond operations, the tool displays a message about a missing guide path and a prompt to convert the affected bond fingers to Free Placement style.

To convert bond fingers to *On Path* wire bonds:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that *Fingers* is selected in the Find Filter.
3. Select the bond fingers, and choose *Create Best Fit Path* from the pop-up menu.
   The tool automatically creates a guide path. Upon the next wire bond move operation, these fingers snap to the newly created guide path.
4. Choose *Done* from the pop-up menu. Otherwise, continue to perform other tasks within this command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding Bond Fingers and Routing Stubs

You can also use the `wirebond add` command to perform this task.

To add bond fingers:

1. Be sure that you set up default wire bond characteristics and wire profiles.
   These default characteristics are used during the bonding operation. However, you can change these characteristics as you add or move wire bonds.
2. Be sure that you have added a bond finger guide path to your design if you are using *Orthogonal* or the *On Path* pattern.
3. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
4. Make sure that you select only *Pins* in the Find Filter.
5. Select all the die pads that should go to the same I/O path. You can also select power and ground pins.
6. Choose *Add Wire Bond* from the pop-up menu.
   The wire bonds appear on your cursor.
7. If necessary, change the parameter settings in the *Options* panel.

   ⚠ If you checked the *Use advanced selection filtering* box in the Wire Bond Settings dialog box (`wirebond settings` command), the Advanced Selection Filtering dialog box appears.

8. To enable the *Routing Stubs* option:
   a. Check *Outer* or *Inner*.
   b. Specify a length for the stubs.
   c. To add a via, click the drop-down list to specify the padstack.

   The tool uses the characteristics as defined in the Wire bond Settings dialog box to bond the pattern. You can change these during design by using the *Options* panel or the right-mouse button *Edit Routing Stubs* menu command. On subsequent Add operations, the latest characteristics set are used as default.
   You can modify routing stubs any time using the cline editing commands under the *Edit* menu (`edit vertex`, `change`, `cline change width`, `spin`, `delete`), or create them manually after the bond fingers have been placed by using the `route radial` command.
   For additional information, see *Routing Stubs* in the *Routing User Guid*e.

9. Drag the cursor.
   As you move the cursor, the locations for the fingers automatically update on the screen, as do any affected bond fingers and bond wires based on the current bubble setting.
   As the tool crosses a power or ground ring, any bond wires matching that net association automatically tack down to the ring, while remaining fingers continue to follow the cursor. Note that the tool ignores the die flag. As you approach a guide path (if you selected *Orthogonal* or *On Path* as a pattern style), the fingers all snap to this path at the necessary spacing. If you selected *Equal Length,* the destination is determined by the length and angle specifications. The cursor determines the destination if you selected *Free Placement*.
10. Click the wire bonds into place.

⚠ Use the `wirebond_center_ring_bonds` environment variable to have the tool run a post-process step to try and center the power and ground wires between the adjacent wires on both sides. This improves the design for the manufacturing stage, but may cause a slight performance decrease during interactive wire bonding. See the *Wirebond* category under *Ic_packaging* in the User Preferences Editor dialog box (`enved` command).

Related Topics

- wbedit
- wirebond settings
- Advanced Selection Filtering Dialog Box
- Wirebond Add Command: Options Panel
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- Advanced Selection Filtering Dialog Box

Editing Routing Stubs

You can also use the wirebond edit command to perform this task.

To edit routing stubs:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Choose *Fingers* in the Find Filter.
3. Choose the finger or fingers on which you want to add, remove, or delete routing stubs.
4. Modify the parameters in the *Options* panel according to the task you are performing.
   The number of new DRC errors based on current settings appears.
5. Click *Update routing stubs*.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding Non-Wired Bond Fingers by Selecting a Bond Finger

You may want to perform this method of adding non-wired bond fingers because the tool uses the same alignment point on the die as the source bond finger.

For additional information, see *Non-Wired Fingers* in the *Routing User Guide*.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that *Fingers* is selected in the Find Filter.
3. Select a bond finger (source) in the design.
4. Choose *Copy – Left/Top* or *Copy – Right/Bottom*.
   Left/Top means that placement of the non-wired bond finger is left of the selected bond finger when copying on either the North or South die sides and on top of the selected bond finger when copying on either the East or the West sides. *Right/Bottom* means that placement of the non-wired bond fingers is right of the selected bond finger when copying on either the North or South die sides and on the bottom of the selected bond finger when copying on either the East or the West sides.
   The *Options* panel displays the wire bond characteristics of the selected finger. A non-wired bond finger with the *On Path* pattern style adheres to the guide path nearest to the cursor. You can see the push and shove effect of the non-wired bond finger on the neighboring bond fingers when you move the cursor.
   The placement for non-wired bond fingers with *Free Placement* or *Equal Wire Length* pattern styles is approximated. You can change the placement for these bond fingers using the *Move* command.
5. Click at the desired location to place the non-wired bond finger.
6. To connect the die to the non-wired bond fingers, see Connecting a Die to Non-wired Bond Fingers below.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding Non-Wired Bond Fingers by Selecting a Die Pad

You may want to perform this method of adding non-wired bond fingers because the tool already aligns them to the pin if you set the Orientation as Aligned with Wire.

For additional information, see *Non-Wired Fingers* in the *Routing User Guide*.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that *Pins* is selected in the Find Filter.
3. Select one or more die pads in the design.
4. Choose *Add* from the pop-up menu.
5. Uncheck *Wire* in the *Options* panel.
   A non-wired finger with the *On Path* pattern style adheres to the guide path nearest to the cursor. You can see the push and shove effect of the non-wired finger on the neighboring bond fingers when you move the cursor.
   For information on the *Options* panel for this command, see Wirebond Add Command: Options Panel.
6. Click at the desired location to place the non-wired bond fingers.
7. To connect the die to the non-wired bond fingers, see Connecting a Die to Non-wired Bond Fingers.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding Non-Wired Bond Fingers by Selecting a Guide Path

You may want to perform this method of adding non-wired bond fingers because the tool aligns the bond finger to the point on the die orthogonal to the place you picked on the guide path.

For additional information, see *Non-Wired Fingers* in the *Routing User Guide*.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that *Lines* is selected in the Find Filter.
3. Select guide path in the design.
4. Right-click, and choose *Create Non-wired Finger*.
   A non-wired finger with the *On Path* pattern style adheres to the guide path nearest to the cursor. You can see the push and shove effect of the non-wired finger on the neighboring fingers when you move the cursor.
   Click at the desired location to place the non-wired finger.
5. To connect the die to the non-wired bond fingers, see Connecting a Die to Non-wired Bond Fingers.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding Non-Standard Wire Bonds

⚠ Use the Wire Bond Edit application mode non-standard wire bond option for pin objects only. For other objects, choose *Route – Wire Bond – Add/Edit Non-Standard*.

Follow these steps to add non-standard wire bonds to your design:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Check *Pins* in the Find Filter, based on the objects you are connecting with the non-standard wire bond.
3. From the pop-up menu of the selected pin, choose *Add Non-Standard Wire Bond*.
4. Complete the parameters in the *Options* pane.
5. Click to specify the end point.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Connecting a Die to Non-wired Bond Fingers

To connect the die to the non-wired fingers created by this command:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that *Pins* is selected in the Find Filter.
3. Select the pins.
4. Choose *Add Wire Bond* from the pop-up menu.
5. Uncheck the *Fingers* box in the *Options* panel.
   The abbreviated *Options* panel for the `wirebond add` command appears. This allows you to add wires from (multiple) die to these fingers, using one or more wire profiles.
   For information on the *Options* panel for this command, see Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers).

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Attaching Multiple Wire Bonds to One Bond Finger

To attach multiple wire bonds to one bond finger, be sure that the die pads to which you are attaching wire bonds are on the same net.

There are two ways to perform this task; the first procedure is faster when adding multiple wire bonds to one bond finger.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Pins* in the Find Filter.
3. Select the die pad to which you are adding a bond wire, and choose *Add Wire Bond* from the pop-up menu.
   The tool adds the wire bond.
4. Select the additional neighboring die pads to which you want to add wires to the bond finger created in step 2.
5. Right-click, and choose *Add* from the pop-up menu.
6. In the *Options* panel, check the *Allow multiple wires to fingers* box.
   The tool places wires from the pads to the specified bond finger.

or

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Pins* in the Find Filter.
3. Select the die pad to which you add a bond wire, right-click, and choose *Add Wire Bond* from the pop-up menu.
   The tool adds the wire bond.
4. Select another die pad on the same net, and choose *Add Wire Bond* from the pop-up menu.
5. In the *Options* panel, check the *Connect wires to existing fingers* box.
   The fields in the *Options* panel change.
6. Check the *Allow connection to wired fingers* box.
   The tool adds the bond wire to the bond finger established in Step 2.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Defining Padstacks for Bond Fingers

When you are adding, moving, or changing bond fingers, you can access the Bond Finger Information dialog box to access another padstack or create a new one. You can also use the wirebond add command to perform this task.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Pins* in the Find Filter.
3. Select all the die pads that should go to the same I/O path. You can also select power and ground pins.
4. Choose *Add Wire Bond* from the pop-up menu.
   The bond wires appear in the design. The *Options* panel of the Control Panel displays the appropriate parameters.
5. In the *Finger* frame, click *Add*.
   The Bond Finger Padstack Information dialog box appears. For information on the dialog box, see Bond Finger Padstack Dialog Box.
6. In the *Method* frame, choose one of the following methods to set the padstack values:
   - *New* to define a new padstack, specifying the dimensions of the die pins instead of using an existing padstack from the design or library.
   - *Load from disk* to import an existing padstack definition from disk.
7. Enter the name to use when the tool creates the new padstack.
8. Enter the substrate layer for the pad in the *Layer* field. The default setting is *SURFACE*
9. Choose an option in the *Shape* frame to use for the pad. The default setting is *Oblong*.
10. Enter the dimensions for the size of the shape for any new pads that you want to define.
11. Click *OK.*
    The tool automatically adds pads on the wire bond layers for connection to the wire bonds.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

WireBonding from a Die to an Interposer

To wire bond from a die to an interposer:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that *Pins* is selected in the Find Filter.
3. Choose the source pins from the upper die component.
4. Choose Add Wire Bond from the pop-up menu.
5. Uncheck the *Finger* check box in the *Options* panel and complete the parameters in the abbreviated *Options* panel.
6. Move your cursor in the direction of the interposer pads.
   The tool wire bonds each die pad to the best destination pad on the interposer, based on proximity to your cursor's position on the screen and an attempt to minimize the crossing of wire bonds.
7. Click to finalize the bond wire pattern.
   At this point, the tool automatically assigns the interposer pads and connections to the net of the die pin to which it is wired. If your ratsnest lines are currently displayed, they should update to reflect the new routing end point for each net off the die.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

WireBonding from an Interposer to the Substrate

To wire bond from an interposer to the substrate:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Ensure that *Fingers* is set in the Find Filter.

   ⚠ This option is not currently available from the `wirebond add` command, as the tool does not enable *Finger* objects in the Find Filter.

   When you select the pads of an interposer for wire bonding, the same pop-up menu is available as if you selected *Pins*. You select *Add Wire Bond*, not the finger manipulation commands such as *Move* and *Delete*. The pads of an interposer are fixed in place and should not be modified (or removed from the design). Therefore, the only applicable action is *Add*.
3. Select the source pads from the interposer symbol.
4. Right-click and choose *Add Wire Bond* from the pop-up menu. Now you are in `wirebond add` mode.
5. Specify the desired wire bond pattern style.
   Typically, this is either *On Path* or *Equal Wire Length*. Configure the remaining fields on the *Options* panel.
6. Move your cursor to the desired location for the reference finger.
   You should see the screen dynamically update the wire bond pattern to match your cursor movements, allowing you to assess when you have reached the ideal placement location.
7. Click to finalize the desired bond wire pattern.
   At this point, the tool automatically assigns the interposer pads and connections to the net of the die pin to which it is wired. If your ratsnest lines are currently displayed, they update to reflect the new routing end point for each net off the die.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Deleting Bond Fingers

You can also use the wirebond edit command to perform this task.

To delete selected bond fingers and any bond wires attached to them:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select the bond fingers that you want to delete, and choose *Delete* from the pop-up menu.
   There is no dynamic push and shove or pattern adjustment during a deletion. This ensures that the rest of the pattern remains uncorrupted by the deletion. You can make any changes to the pattern using the `move` command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Moving Bond Fingers

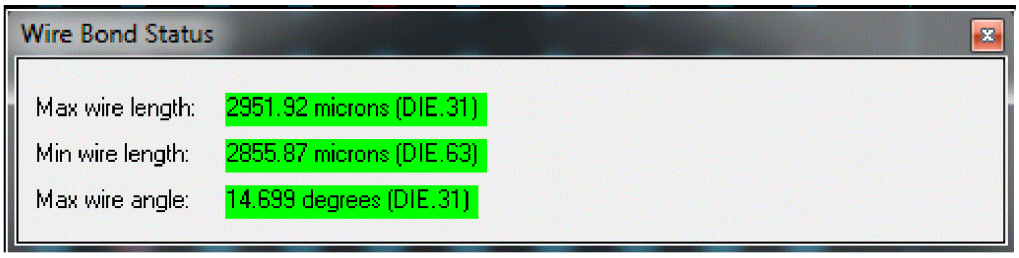
⚠ Use the `wirebond_center_ring_bonds` environment variable to have the tool run a post-process step to try and center the power and ground wires between the adjacent wires on both sides. This improves the design for the manufacturing stage, but may cause a slight performance decrease during interactive wire bonding. See the *Wirebond* category under *Ic_packaging* in the User Preferences Editor dialog box (`enved` command).

You can also use the wirebond edit command to perform this task.

To manipulate bond fingers and any bond wires along the existing finger guide paths:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select the bond fingers and bond wires that you want to move, and choose *Move* from the pop-menu. By default, if two wires use different profiles, the tool allows them to cross within a set distance from the die or pin. To avoid the crossing of wires in your design during pushing and shoving of bond fingers (such as when optimizing the design for optical checking), set the WIREBOND_IGNORE_WIRE_PROFILES environment variable in the User Preferences (`enved` command).
   Setting the WIREBOND_IGNORE_WIRE_PROFILES environment variable causes the tool to treat all wires as part of the same profile for placement only. DRC computations, 3D views and 3d analysis are unaffected by this setting.

   ✓ The Wire Bond Status form appears, displaying the *Max wire length*, *Min wire length*, and the *Max wire angle*. You can change the information displayed in the form by setting *wirebond_hud_line_1*, *wirebond_hud_line_1*, and *wirebond_hud_line_1* under *Ic_packaging – Wirebond* in User Preferences Editor.



4. Check the current bubble mode in the *Options* panel. The fingers that you selected maintain their relative spacing as you move them. For a description of these fields, see Wirebond Add Command: Options Panel.

5. Slide the cursor to the desired position, keeping an eye on the Wire Bond Heads-up Display and on the affect of the movement on the rest of the fingers in your pattern.
   Affected wire bond pads bubble out of the way, based on your settings.
6. To move the selected fingers to a different guide path, drag the cursor to that guide path. When you have adjusted the pattern to your satisfaction, click the mouse to finalize the placement.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- enved

Swapping Bond Fingers

You can also use the wirebond edit command to perform this task.

To swap the locations of any bond fingers for which the bond wires cross:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select two or more bond fingers and bond wires to swap, choose *Swap* from the pop-up menu.
   The tool recursively swaps items until there are no crossed bond wires. If you select elements from multiple paths, the tool maps the swap locations onto the same guide path as the original finger. Therefore, a bond finger does not move from one path to another during a swap, but rather swaps along its path to the best estimated location based on the other point.
   This task does not adjust the pattern if DRC errors result.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Spacing Bond Fingers Evenly

You can also use the wirebond edit command to perform this task.

To space selected bond fingers evenly, be sure that the selected fingers all lie on the same guide path:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select three or more bond fingers that you want to space evenly, and choose *Space Evenly* from the pop-up menu.
   The first and last fingers retain their current positions. All fingers in between adjust to have the same spacing between each pair of neighboring items. The tool performs this action regardless of any DRC errors that may result from spacing in other areas of the pattern.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adjusting Bond Fingers to Meet Minimum DRC Requirements

You can also use the wirebond edit command to perform this task.

To spread or compress bond fingers to meet minimum DRC requirements:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select two or more fingers to adjust. The reference finger needs to be one of the selected fingers.
4. Choose *Adjust Min DRC* from the pop-up menu.
5. Select the reference finger for the operation, which remains in its current location after the adjustments.
   At this point, the tool either spaces the remaining fingers farther apart or closer together, depending on the existing element spacing. This action takes place regardless of any DRC errors that may result from spacing in other areas of the pattern.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Centering Bond Wires Between Bond Fingers

You can also use the wirebond edit command to perform this task.

This task is ideal for assembly considerations, as it maximizes the tolerance for both the wire bonding machine and the possible bending of bond wires. Be sure that you have two guide paths with bond fingers in your design.

To center bond wires between bond fingers so that the fingers on each side are an equal distance away:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select the bond fingers, and choose *Center Wires* from the pop-up menu.
   In cases where two or more wire bonds exist between the two adjacent fingers, the tool computes the total available gap between fingers and the space is divided equally between each gap.

⚠ Set the `wirebond_center_same_profile` environment variable (`enved` command) so that the tool ignores wires on different profiles and looks for the nearest adjacent wire on the same profile as the wire being moved. The default behavior is to use the next wire bond found on any wire profile.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adding or Deleting Routing Channels

You can also use the wirebond edit command to perform these tasks.

To add or delete routing channels between bond fingers:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select two bond fingers, and choose *Add Routing Channel(s)* or *Delete Routing Channel(s)* from the pop-up menu.
   The Routing Channel Dialog Box appears.
4. If you are adding routing channels, complete the parameters and click *Add*.
5. When this message appears in the console window:
   Select reference wire bond. Note that reference wire bond must be on a guide path and not freely placed
   click the reference wire bond.
   The tool calculates the spacing, or removes the channels.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Creating a Merged Finger Shape

You can also use the wirebond edit command to perform this task.

You can select two or more bond fingers on the same net and move them as a group during push and shove operations.

To merge neighboring bond fingers on the same net:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Fingers* in the Find Filter.
3. Select two bond fingers, and choose *Create Merge Finger Shape* from the pop-up menu.
   The tool lays a conducting shape on top of the fingers. This shape establishes electrical connections between the two fingers, and makes it appear to a bonding tool as a single finger with multiple connection points. The tool maintains the shape over fingers as you move them as a group during push and shove operations.

   ⚠ The two original finger padstacks remain in the database.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Removing a Merged Finger Shape

You can also use the wirebond edit command to perform this task.

To remove the shape added to a set of bond fingers when you selected the *Merge Finger Shape* menu item, and restore the set of original fingers:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Be sure to check *Fingers* in the Find Filter.
3. Select the merged fingers, then right-click, and choose *Remove Merge Finger Shape* from the pop-up menu.
   The tool removes the shape overlaying the two fingers, making them separate entities.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Creating a Ring Segment from a Set of Bond Fingers

You can also use the wirebond edit command to create a ring segment from a set of placed bond fingers.

To create a ring segment from a set of placed bond fingers:

1. From the `wirebond select` command, make sure that you select only *Fingers* in the Find Filter.
2. Select the bond fingers, then right-click, and choose *Advanced – Create Ring* from the menu.
   The *Options* panel displays the appropriate parameters.
3. Complete the values and click *Generate*.
   The bond fingers convert into a segment of a ring with the same assigned net. Ring tack connections replace the original finger padstacks.
4. After using the `wirebond select` command, right-click and choose *Done* from the pop-up menu.
   Otherwise, continue to perform other tasks within this command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Changing the Characteristics of Wire Bonds

You can select a set of wire bonds and assign them common characteristics. The characteristics of the wire bonds do not have to be the same initially.

You can also use the `wirebond edit` command to change the characteristics of the bond fingers.

To change the characteristics of the bond fingers:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select either *Fingers* or *Bond wires* in the Find Filter.
3. Select the bond fingers or wires, and choose *Change Characteristics* from the pop-up menu. The *Options* panel displays the appropriate parameters you can change based on whether you selected *Fingers* or *Bond wires* in the Find Filter.
4. Modify the values and click *Apply changes and exit.*

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- Wirebond Select Command - Change Characteristics Command: Options Panel

Redistributing Bond Fingers Across Multiple Bond Finger Guide Paths

You can also perform this task using the `wirebond manage guide paths` command.

To redistribute bond fingers across multiple bond finger guide paths:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you check *Lines* in the Find Filter.
   The *Options* panel displays parameters.
3. Modify the parameters if necessary.
4. Select the bond finger guide paths from the fingers' current location.
5. Choose *Redistribute Fingers* from the pop-up menu.
   This message appears:

   Select path(s) to split to or to merge to.
6. Window around the path with bond fingers and the destination path.
   The bond fingers redistribute across the selected paths.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- Wirebond Mange Guide Paths - Redistribute Fingers Command: Options Panel

Adding a Jumper between Bond Fingers

You can also use the `wirebond edit` command to add a jumper between bond fingers.

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you check *Fingers* in the Find Filter.
3. Select two bond fingers that are on the same net.
4. Choose *Add Jumper* from the pop-up menu.
   The *Options* panel displays parameters. For information on these fields, see .
5. Modify the parameters if necessary.
6. Click in the Design Window to display the jumper.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- Wirebond Select Command - Add Jumper Command - Options Panel

Populating a Guide Path with Bond Fingers

Use this /feasibility command early in your wire bond design flow to determine how many bond fingers you can place on the guide path, given your specified settings and optimal conditions.

1. Place one or more die in your design.

   ⚠ This step is optional. You can also perform this task without a die.

2. Add linear or arc wire bond guide paths at the desired distance.
   If no die exists, use the `add line` or `add arc` command to add guide paths on the WB_GUIDE_LINE layer.
3. Run the `wbedit` (or `wirebond manage guide paths`) command.
4. Select the guide paths that you created in Step 2, choose *Populate with Fingers* from the pop-up menu.

   ⚠ The guide path that you select for populating with fingers must be empty.

5. Modify the fields in the *Options* panel as required to specify bond finger-to-bond finger spacing, the padstack for use, and so on.
   As you change each field, the tool updates the display and indicates a count of bond fingers can be placed on each guide path.
   The tool places the bond fingers from the end of the guide path.
   Because there are no wires with this command, if you set *Orientation* to *Aligned With Wire* in your wire bond settings, the tool aligns all the fingers to point to the center of the die. If there is no die present, the tool aligns all the fingers to point to the center of the design.
6. When you are satisfied with the placement, click *Create fingers* to save the bond fingers to the design.
7. When you are ready to connect the die to the bond fingers:
   a. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
   b. Select pins and choose *Add Wire Bond* from the pop-up menu.
   c. Check the *Connect wires to existing fingers* box in the *Options* panel.
      This allows you to add wires from (multiple) dies to these fingers, using one or more wire profiles.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Deleting Bond Wires

You can also use the wirebond edit command to perform this task.

To delete selected bond wires:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Bond wires* in the Find Filter.
3. Select the bond wires delete, and choose *Delete* from the pop-up menu.
   A confirmation box appears asking if you want to delete the bond fingers also.
4. Click *Yes* to delete the fingers or click *No* to delete the bond wires only.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Moving, Centering, or Swapping Bond Wires

You can also use the `wirebond edit` command to perform these tasks.

To move, center, or swap bond wires:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Bond wires* in the Find Filter.
3. Select the bond wires to move center, or swap, then choose *Move, Center,* or *Swap* from the pop-up menu.
   If you are swapping or centering bond wires, a message appears in the console window indicating which bond wires were swapped or centered.
4. If you are moving bond wires, when you reach your destination, click the mouse.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Adjusting Bond Wires to Meet Minimum DRC Requirements

You can also use the wirebond edit command to perform this task.

To adjust bond wires to meet the minimum DRC requirements:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Make sure that you select only *Bond wires* in the Find Filter.
3. Select the bond wires to adjust, and choose *Adjust to Min DRC* from the pop-up menu.
4. Select the reference finger for the operation.

The tool spaces the remaining wires farther apart or close together, depending on the existing wire spacing. The tool performs this action regardless of the DRC errors that may result from the spacing in other areas of the pattern.

5. After using the `wirebond select` command, right-click and choose *Done from* the pop-up menu. Otherwise, continue to perform other tasks within this command.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Splitting or Reconnecting Power/Ground Ring

Follow these steps to split or reconnect power/ground ring:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Select *Shapes* in the Find Filter.
3. Select the ring that you are going to segment or reconnect.
4. Choose *Cut* or *Heal* from the pop-up menu.
   A message appears:

   Define the area of the shape to modify.
5. If you are splitting the ring, use the cursor to draw a rectangle, circle, or polygon on the screen, and then click to complete the operation.
   The tool removes any piece of the shape within that rectangle.
   Both sections of the split ring retain the same net assignment. You can reassign the net using the *Logic – Assign Net* menu command or other *Logic* commands.
6. If you are reconnecting the ring segments, use the cursor to draw a rectangle or circle encompassing the two ends of the shapes to reconnect on the screen. Click to complete the operation.
   The tool recreates any piece of the original shape that lies within that windowed area and merges any overlapping sections into a single shape.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Assigning Nets to a Ring or Ring Segment

You can temporarily short a set of nets to a power or ground ring before you cut the ring into multiple segments, or assign a different net to an existing ring segment.

To assign nets to a ring or ring segment:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Select *Shapes* in the Find Filter.
3. Select two shapes on a power/ground net or a dummy net.
   You cannot select a shape on a signal net.
4. Choose *Assign Nets* from the pop-up menu.
   The Assign Nets dialog box appears. See Assign Nets Dialog Box.
5. Use this dialog box to change the master net name if desired, and add shorted nets to the master net. Click *OK*.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Editing Ring Parameters for a Power/Ground Ring

You can change the ring parameters:

1. Ensure that the application mode is set to Wire Bond Edit (*Setup – Application Mode – Wire Bond Edit*).
2. Select *Shapes* in the Find Filter.
3. Choose *Edit Parameters* from the pop-up menu of the selected ring.

   ⚠ When you change settings for a split ring, all pieces of the original ring will be updated with the changes.

4. Specify the distance and width.

   ⚠ The individual soldermask openings are not affected by editing the ring. However, if you create soldermask openings in the padstack definition for the ring tack point finger's padstack, the individual mask openings for each tack point will be updated on editing the ring.

5. Click *Update ring*.

Related Topics

- wbedit
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- wbedit Command: Options Panels
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

window center

The `window center` command lets you choose the location that appears in the center of the display window.

Selecting the Location in the Center of the Window

To choose the location in the center of the design window, follow these steps:

1. Run `window center`. The following message displays:

   Pick Center For New Display.

2. Position the cursor at the location for the center of the window and click.

window fit

The `window fit` command adjusts the magnification of the display (whether zooming in or out) to fit your design window.

window in

The `window in` command magnifies the design around the location of the cursor. You can continue to zoom in on a design by repeating the command.

window out

The `window out` command reduces the magnification of the design around the location of the cursor. You can continue to zoom out on a design by repeating the command.

window points

The `window points` command specifies the outer boundaries of the display window. You can use it to magnify a specific area of a design as well.

Specifying an Area of a Design

Perform these steps to specify the boundaries of the display window:

1. Run `window points`. The program displays the message:

   Pick 1st Corner Of New Window.

2. Position the cursor and click. The program displays the message:

   Pick To Complete Window.

3. Move the cursor to position a rectangle that represents the outer boundary the display window.
4. Click the mouse. The program displays the area you have identified in the display window.

window world

The `window world` command controls what area of your layout displays and to zoom into (display magnified) and zoom out from the layout. Changes the display in the Design Window to match the display shown in the World Window. This is a convenient method for quickly reducing the magnification used to display a design.

wirebond add

The `wirebond add` command lets you add wire bonds to your design. You can also access existing bond finger padstacks or create new ones when running this command. Be sure that you first create a bond finger guide path (wirebond select command) unless you are adding die-to-die wire bonds, creating an *Equal Wire Length* pattern, or using the *Free Placement* option. With the `wirebond add` command, you can also connect die pads to to existing non-wired bond fingers.

⚠ By default, the tool does not let you add multiple wire bonds to a pin. To override this behavior, set the `wirebond_multwire_pins` environment variable and connect multiple wire bonds to a single pin. You can set this variable by choosing *Setup – Preferences* (`enved` command), then clicking the *IC_packaging* category and then *Wirebond*.

In addition to using this command, you can use the wirebond select command to easily perform these tasks.

For additional information on flows and wire bond tools, see the *Routing User Guide* in your documentation set. For information on design tasks, see Advanced Selection Filtering Dialog Box.

Related Topics

- Wirebond Add Command: Options Panel
- Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers)
- Bond Finger Padstack Dialog Box
- Wirebond Add Procedures
- Advanced Selection Filtering Dialog Box

Wirebond Add Command: Options Panel

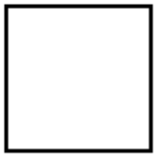
Access Using

- *Menu Path: Route – Wire Bond – Add*
- Toolbar Icon: 

The three modes of wire bonding determine how you check the boxes on the *Options* panel and how you proceed.

Table 1.1: Modes of Wire Bonding Operation

If You Are Performing....	Wire Check Box	Finger Check Box	Then...
Typical wire bonding (adding bond wires and fingers)	☑	☑	after you check the boxes, complete the *Options* panel.
Reconnection or connection to existing bond fingers, interposer pads, or die pads	☑	☐	after you change to *Wire* mode, complete the Abbreviated *Options* panel that appears. ⚠ Be sure to uncheck the *Add to Finger Group* box if you want the bond fingers to belong to a different group.

Creation of bond fingers only			after you change to *Finger* mode, complete the *Options* panel.

The *Options* panel displays the following parameters when you are adding or moving bond fingers.

Add wire bond	
Wire	When you check this box, the tool creates the wire portion of a wire bond. To add wires to existing bond fingers or pins, check this box and uncheck the *Fingers* box. An Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers) appears.
Finger	When you check this box, the tool creates the bond finger portion of a wire bond. To add non-wired bond fingers, check this box and uncheck the *Wires* box. Non-wired bond fingers push and shove nearby wired bond fingers in the same manner as if non-wired bond fingers were connected to a die pad.
Bubble	Specifies the method for pushing or shoving existing bond fingers and bond wires. Once you choose a method, it is saved as the default with the design even when the current layout session ends and you start a new one. You can choose: • *Shove all* – Pushes and shoves all items necessary based on cursor movements. This is the default setting. • *Shove path* – Pushes and shoves only those items on the same path as the selected items. • *Shove Off* – Does not push or shove anything not selected for movement. It updates the heads-up display showing the state of the minimum and maximum wire length and wire angle values.
Pattern	

Style	Specifies the wire bond pattern. Choices are: • *Orthogonal – Specifies a pattern of* 0-degree bond wire angles and bond pads. • *Equal Wire Length* – Specifies that wire bond angles are variable, but all wire bond lengths are equal. • *On-Path* – Specifies that all wire bonds stay on the bond finger guide path. • *Free Placement* – Does not snap to any path. With this option, the fingers are never automatically pushed or shoved due to bubbling as they have no associated path of movement. This field determines the placement rules applied based on the path that these elements follow. The default setting is *On-Path unless there are no guide paths; then the default setting is Equal Wire Length*.
Length	Specifies the wire length. This field is available only if you select *Equal Wire Length* as a style. ⚠ For non-wired bond fingers, this length is approximated for the placement of non-wired bond fingers since there is no measurable length of wire from pin to bond finger. For non-wired bond fingers that have the *On Path* pattern but have no paths because they were removed after the group was created, the placement is approximate and not based on the value in the *Length* field since there is no path to snap to.
Bond voltage nets to rings	Specifies that the tool bonds voltage nets to the power and ground rings they cross rather than to bond fingers. This enhancement is for stacked dies and multi-row single dies with internal power signals that should go to the bond fingers and not the rings. Uncheck this box if you have this type of design. The default setting is checked. If you are adding wires only, this field is not available.
Wire	
Allow multiple wires to finger	Check this box to allow the tool to add multiple bond wires to a bond finger.

Profile	Specifies the wire bond profile (including diameter) to associate with this wire. A profile determines the approximate 3D path that the wire bond follows. To specify the name of a new profile, enter the name in this field and then use the *Route – Wirebond – Settings – View/Edit wire profiles* to create the profile in the profile editor. Then you can select the profile from the pull-down menu. Any number of groups can reference the same wire profile, but only one wire profile can be referenced by each group.
Finger	
Pad	Lists the available padstacks that you select for the bond fingers. The default padstack is specified in the Wire Bond Settings dialog box (see the wirebond settings command).
Add	Displays the Finger Padstack dialog box which allows you to define a new finger padstack.
Align	Specifies the orientation to use when placing bond fingers. Choices are: • *Aligned with Wire* • *Orthogonal to Die Side* • *Orthogonal to Guide* • *Pivoting Ortho to Guide* • *Average Wire Angle* • *Constant Angle* • *Match CW Neighbor* • *Match CCW Neighbor* The default setting is *Aligned with Wire*. Choose the *Pivoting Ortho to Guide* option for oblong or rectangular bond fingers. It lets you avoid the situation when the wire is almost perpendicular to the die side and the bond wire crosses the boundary of the bond finger over the rounded end of the bond finger. As the bond wire angle increases, the bond wire starts to cross the bond finger boundary on one of its straight edges. With this option, the tool pivots the bond finger slightly so that the bond wire crosses over the rounded end.
Angle	

Snap	Controls which point on the bond finger snaps to the guide path. Because bond fingers vary in size, sometimes it is necessary to control how they follow the path so that bond wire lengths are equal and you achieve the smoothest possible pattern. Options are: • *Center of Finger* • *Finger Origin* • *Near End* (the tip of the bond finger closest to the die edge) • *Far End* (the tip of the bond finger farthest from the die edge) • *Nearest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is closest to the die) • *Farthest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is farthest from the die) The default setting is *Finger Origin*.
Routing Stubs	
Outer	Check this box to create a stub from the bond finger in the direction away from the die edge. The default setting is *Off*, which disables the *Length* and *Padstack* fields.
Inner	Check this box to create a stub from the bond finger in the direction toward the die edge.
Outer / Inner Length	Specifies the length of the stub, which is measured from the end of the bond finger from which the stub exits. By default, it is twice the width and length of the default bond finger padstack to ensure that there is clearance from the bond finger. The tool always aligns the stubs with the bond finger's rotation to ensure a good routing from the die for routing and manufacturing.
Outer /Inner Via Pad	Specifies the padstack to use for the via at the end of the routing stub, if desired. By default, this field is set to <*No Via*>, and no via is added.

Related Topics

- Bond Finger Padstack Dialog Box
- Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers)
- Wirebond Add Procedures

Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers)

When you want to connect wires from pins to existing non-wired bond fingers and you check only the *Wires* field (uncheck the *Fingers* field) in the *Options* panel during the `wirebond add` command, an abbreviated *Options* panel displays the following parameters.

⚠ In other scenarios where you are working with different die variants or are importing saved bond finger pattern styles, this option lets you connect the pins of a particular die or die stack to the imported bond fingers.

Add wire bond	
Wire	When you check this box, the tool creates the wire portion of a wire bond.
Finger	When you check this box, the tool creates the bond finger portion of a wire bond.
Destinations	
Connect to die pads	Lets you create direct pad-to-pad wire bonds when the pads are on the same nets.
Same side	Lets you connect only to bond fingers on the same side as the reference pin's mapped bond finger.
Same guide path	Lets you connect only to bond fingers on the same side and the same path as the reference pin's current destination bond finger.
Wire	
Allow multiple wires to finger	Specifies that the finger already wired is considered a legal destination to which selected pins can connect. Usually, the tool allows only one bond wire to connect to each bond finger. To allow this action to occur, the net on the finger and pin must match. By default, this box is not checked.

Profile	List of profiles.
Finger	
Adjust finger rotation	Adjusts the finger rotation based on the orientation of the wire to which it connects. This action is based on the *Orientation* setting that you specified in the Wire Bond Settings dialog box. By default, this box is not checked.

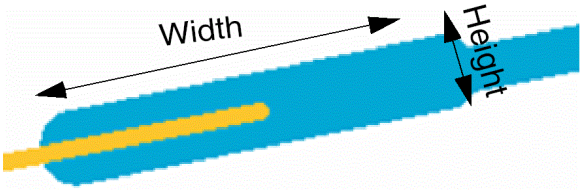
Related Topics

- Advanced Selection Filtering Dialog Box
- Wirebond Add Procedures

Bond Finger Padstack Dialog Box

This dialog box appears when you click *Add* in the *Options* panel when running the `wirebond add` command. Be sure that you configure the correct length and width for the bond finger pads or they will not display correctly.

Method	
New	Defines a new padstack, specifying the dimensions of the die pins instead of using an existing padstack from the design or library.
Load from Disk	Indicates that you can import an external padstack definition. Clicking *Browse* lets you locate the padstack on your disk. When you import the padstack, the *Specifications* frame reflects the padstack information. You cannot edit the padstack specifications.
Specifications	
Name	Specifies the name to use when the tool creates the new padstack.
Layer	Specifies the substrate layer for the pad. The default is the top conductor layer in the design.
Padstack	
Shape	
Oblong	Specifies that the tool use an oblong as the padstack shape. The default setting is *Oblong*.
Rectangle	Specifies that the tool use a rectangle as the padstack shape.
Dimensions	

Width	Specifies the width for an oblong or rectangle for any new padstack that you want to define. Be sure that the value for this field is greater than the value for the *Height* field for proper finger rotation and orientation. Note that the long axis of the finger is the width. 
Height	Specifies the height for an oblong or rectangle for any new padstack that you want to define.
OK	Accepts the entry and closes the dialog box.
Cancel	Cancels the operation and closes the dialog box.
Help	Displays the help window for the dialog box.

Related Topics

- Advanced Selection Filtering Dialog Box
- Wirebond Add Command: Options Panel

Wirebond Add Procedures

You can use the `wirebond add` command to perform the following procedures but it is generally easier to use the `wirebond select` command, especially if you are performing several tasks. The procedures are described under `wirebond select`. See:

- Adding Bond Fingers and Routing Stubs
- Defining Padstacks for Bond Fingers
- Adding Non-Wired Bond Fingers by Selecting a Bond Finger
- Adding Non-Wired Bond Fingers by Selecting a Die Pad
- Adding Non-Wired Bond Fingers by Selecting a Guide Path
- Connecting a Die to Non-wired Bond Fingers

You cannot use the `wirebond add` command to perform these tasks. For these procedures, see:

- WireBonding from a Die to an Interposer
- WireBonding from an Interposer to the Substrate

Related Topics

- Advanced Selection Filtering Dialog Box
- Wirebond Add Command: Options Panel
- Abbreviated Options Panel for the Wirebond Add Command (Connecting Die Pads to Bond Fingers)

wirebond add nonstandard

The `wirebond add nonstandard` command lets you add non-standard wire bonds and arrays of bond wires for high-current devices and redundancy between two objects.

Related Topics

- Adding Non-standard Wire Bonds
- Non-Standard Bond Wires

Wirebond Add Nonstandard Command: Options Panel

Access Using

- *Menu Path: Route – Wire Bond – Add/Edit Non-Standard*

Short wire to different net	Select to short fingers using a bond wire.
Snap to pad origin	If you check this box, when you select a padstack-based object as either the start or end of the wire, the tool snaps the wire end to the pad origin location. If you do not enable this option, the tool connects to the location on the pad where you click. The default setting is *On*.
Snap to shape center cline	When connecting to a cline snap to shape center.
Profile	Specifies the wire profile to use. You can select a profile from the drop-down list or enter a new name. This setting defaults to the default profile specified in the `wirebond settings` command or *Nonstandard* if there are no available profiles.
Padstack for cline connections	Select the (single-layer) finger padstack to use when connecting wire directly to a cline end.
Multi-wire pattern	Select to create arrays of wires. Enables the Option and value table to specify settings for the wire array. Not selected by default.
Rows	Specifies the number of rows of bond wires in the multi-wire array pattern.
Columns	Specifies the number of wires in a row.
Row spacing	Specifies the spacing between the rows.
Column spacing	Specifies the spacing between the wires in a row.

⚠ For the Start X/Y Offset and End X/Y Offset, for the origin, this is the offset to the middle of the pattern of wires, while for East, West, North and so on it is from the edge of the pad to the easternmost, westernmost, or northern most wire and so on.

Start X Ref	Specifies the reference point of the start of the array for the X axis of the pad. The values are *Center* (default), *Origin*, *East Edge*, and *West Edge*.
Start X Offset	Specifies the offset from the starting X reference. Default value is `0.00` UM.
Start Y Ref	Specifies the reference point of the start of the array for the Y axis of the pad. The values are *Center* (default), *Origin*, *North Edge*, and *South Edge*.
Start Y Offset	Specifies the offset from the Y start reference. Default value is `0.00` UM.
End X Ref	Specifies the reference point of the end of the array for the X axis of the pad. The values are *Center* (default), *Origin*, *East Edge*, and *West Edge*.
End X Offset	Specifies the offset from the ending X reference. Default value is `0.00` UM.
End Y Ref	Specifies the reference point of the end of the array for the Y axis of the pad. The values are *Center* (default), *Origin*, *North Edge*, and *South Edge*.
End Y Offset	Specifies the offset from the ending Y reference. Default value is `0.00` UM.

wirebond edit

The `wirebond edit` command lets you perform a variety of tasks such as manipulating bond fingers and wires, creating rings, adding and deleting routing channels, and creating or removing merge finger shapes. You can also use the pop-up menu to change wire bonding settings.

> ⚠ For many wire bond tasks, the *Options* panel lets you select a wire bond profile. If you enter a new name instead of selecting a listed profile, a new profile with default values is created for you.

In addition to using this command, you can use the wirebond select command to easily perform these tasks.

For additional information on flows and wire bond tools, see the *Routing User Guide* in your documentation set. For information on design tasks, see Advanced Selection Filtering Dialog Box.

Related Topics

- Add Routing Channel Dialog Box
- Wirebond Edit Procedures

Wirebond Edit Command - Move Command: Options Panel

Access Using

- *Menu Path: Route – Wire Bond – Edit*
- Toolbar Icon: 

For descriptions of the fields in the *Options* panel, see Wirebond Add Command: Options Panel.

Related Topics

- Wirebond Edit Procedures

Add Routing Channel Dialog Box

This dialog box appears when you use the `wirebond edit` command with the *Routing – Add Routing Channels* menu item. You can also use the `wirebond select` command to add routing channels.

Number of Routing Channels	Specifies the number of routing channels (a combination of cline width, cline-to-pin and cline-to-cline spacing on the CONDUCTOR layer) that you want added between the two bond fingers.
Exact Spacing	Specifies a numeric value. This value is not affected if you change any of the constraint values.
Add	Completes the operation with the settings specified.
Cancel	Cancels the operation.
Help	Displays help for this dialog box.

Related Topics

- wirebond edit

Wirebond Edit Procedures

You can use the `wirebond edit` command to perform the following procedures but it is generally easier to use the `wirebond select` command, especially if you are performing multiple tasks. The procedures are described under `wirebond select`t command. See:

- Editing Routing Stubs
- Deleting Bond Fingers
- Moving Bond Fingers
- Swapping Bond Fingers
- Spacing Bond Fingers Evenly
- Adjusting Bond Wires to Meet Minimum DRC Requirements
- Centering Bond Wires Between Bond Fingers
- Adding or Deleting Routing Channels

After using the `wirebond select` command, right-click and choose *Done* from the pop-up menu. Otherwise, continue to perform other tasks within this command.

- Creating a Merged Finger Shape
- Removing a Merged Finger Shape
- Creating a Ring Segment from a Set of Bond Fingers
- Deleting Bond Fingers
- Moving, Centering, or Swapping Bond Wires
- Adjusting Bond Fingers to Meet Minimum DRC Requirements
- Splitting or Reconnecting Power/Ground Ring

Related Topics

- wirebond edit
- Wirebond Edit Command - Move Command: Options Panel

wire bond escape

Use the `wire bond escape` command to perform die escape routing after you complete wire bonding a design and before you perform routing tasks.

Related Topics

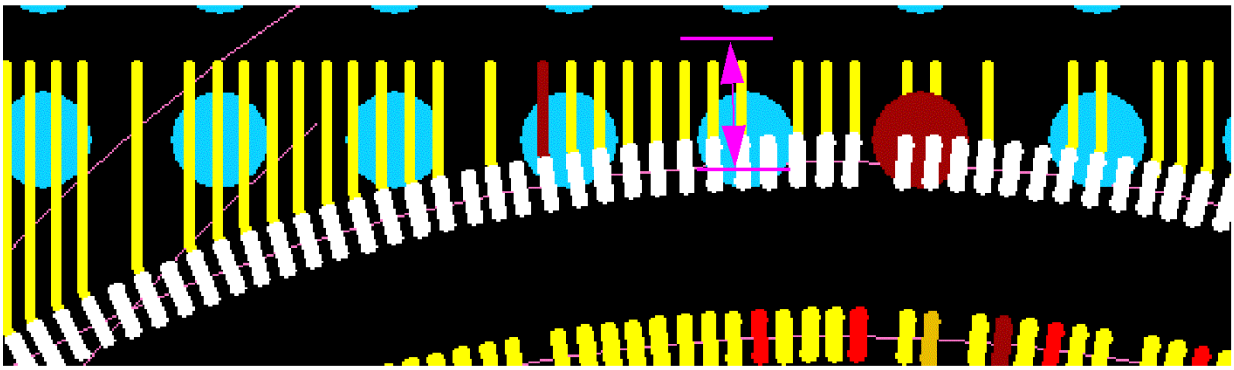
- Wire Bond Escape: Main Pop-up Menu
- Die Escape: Escape Direction Dialog Box
- Escaping from a Wire Bond Die

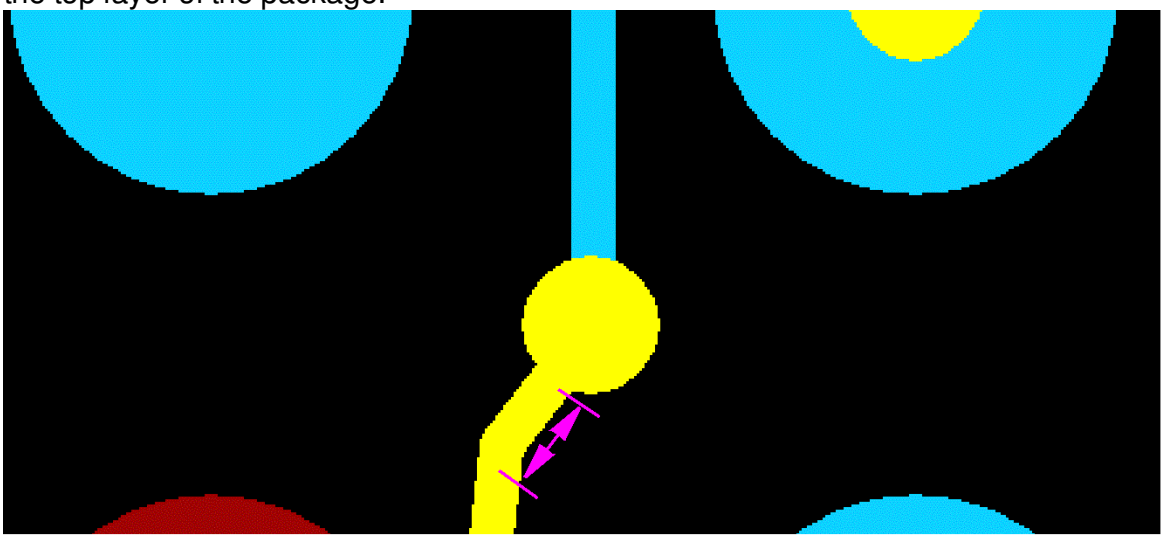
Wirebond Escape Command: Options Panel

Access Using

- *Menu Path: Route – Wire Bond Die Escape Generator*

When you run the wire bond escape command, the *Options* panel of the Control Panel displays these parameters.

Generate Escapes for Voltage Nets	If you check this box, the tool creates escape clines for voltage vias and bond fingers. Voltage vias and bond fingers are those that belong to a net with the VOLTAGE property or its die pin has the PINUSE property set to Power or Ground. By default, this box is not checked.
Distance To Escape From Outer-Most Bond Fingers	Defines the distance from the bond fingers to the location where you want the escape clines to terminate. Once the tool generates escapes, all the escape clines on all sides of the die appear in a horizontal or vertical line at this position, thus defining the escape perimeter. By default, this value is 0. Enter the value as the units of the package. 

Min Bend Distance From Escaping Bond Finger	Specifies the minimum distance of the clines extending from the bond fingers to the bend point. This value defaults to four times the default *Line Width* for the top layer of the package. 
Min Bend Distance From Escaping Via Pad	Specifies the value from the bend point created by the tool to the via pad. Specifying this value helps to avoid acid traps. The value defaults to one-half of the default *Line Width for* the top layer of the package. 

Options for Add Via To Bond Finger	The following parameters apply when you are in the *Add Via to Bond Finger* mode.
Finger	Choose the subclass of the bond fingers.
Escape	Choose the subclass where the escape clines are created.
Padstack	Choose a padstack from the valid list of padstacks that apply to the specified subclasses.
Direction to advance after Add Via • *North* • *South* • *East* • *West*	These buttons are available only when you drag a via. At that time, two buttons are available: either *North* and *South* or *East* and *West*. These buttons determine which bond finger the tool should select next after you drop the current via. For example, if you are on the north side of the die, the options for *East* and *West* are available. If you choose *East*, the tool selects the next available bond finger to the right of the current via. The tool's default direction advances you away from the center of the die, although you can use these radio buttons to override the default. Moving away from the die center encourages you to start in center, which is the densest region of the package, and then move toward the less dense regions at the package corners.

Related Topics

- Die Escape: Escape Direction Dialog Box
- Escaping from a Wire Bond Die

Wire Bond Escape: Main Pop-up Menu

When you are in the `wire bond escape` command and you right-click in the design, the following pop-up menu appears.

Done	Choose this option to commit any work that you have done and exit the `wire bond escape` command.
Oops	Choose this option to undo the last database change.
Cancel	Choose this option to exit the `wire bond escape` command after reverting the database to the state that it was in before you performed any die escapes.
Temp Group	Choose this option to fine tune your selection of elements.
Reject	Available only when you select the *Temp Group* option. Choose this option to undo your last selection.
Deselect Successfully-Escaped Bond Fingers/Vias	Deselects any bond fingers or vias whose escape clines have successfully reached the escape perimeter.
Assign Escape Direction	Causes the Die Escape: Escape Direction dialog box to appear. You do not have to complete this dialog box if the vias and bond fingers already escaped in the direction that you want.
Generate Escapes	Generates an escape cline for each selected bond finger.
Delete Escapes	Deletes the escape clines for selected bond fingers. The escape cline must have one end connected either to a bond finger or a via, and the other end must be unconnected. This option is useful for deleting unsuccessful escapes.
Add Via To Bond Finger	Use this option when escaping from the die on another layer. A via appears and dynamically tracks the cursor. The tool assigns the padstack that you selected in the *Options* panel and automatically generates two clines. One is between the bond finger and the via (on the *Finger* subclass), and a second cline is between the via and the escape perimeter (on the *Escape* subclass). As the cursor moves, the via moves, and the two clines are updated dynamically. When you select a bond finger in a differential pair, the tool highlights the associated differential pair network. The pop-up menu changes when you are in *Add Via To Bond Finger* mode.
Options	

Generate Escapes for Voltage Nets	Choosing this option is the same as checking the corresponding box in the *Options* panel. If you choose this option, the tool creates escape clines for voltage vias and bond fingers. Voltage vias and bond fingers are those belonging to a net with the VOLTAGE property or its die pin has the PINUSE property set to Power or Ground.

Related Topics

- wire bond escape
- Escaping from a Wire Bond Die
- Die Escape: Escape Direction Dialog Box

Die Escape: Escape Direction Dialog Box

This dialog box appears when you choose *Assign Escape Direction* from the main pop-up menu during the `wire bond escape` command.

North South East West Nearest Edge of Die	Click one of the five radio buttons to specify an escape direction.
Assign Escape Direction	Click to assign the selected setting to the corresponding die pin for each selected bond finger or via. By default, vias and bond fingers escape to the nearest edge of the die.
Cancel	Click to close the dialog box without changing the database.
Help	Click to display context-sensitive help.

Pop-up Menu During Add Via To Bond Finger Mode

When you add vias and you right-click, the pop-up menu displays the menu options described below.

Add Via and Advance (single-click)	Choose this option from the pop-up menu (or click the mouse in the design) once you position the dynamic via with the cursor. The tool drops the via and its two new clines into position. It automatically selects the next bond finger (following the direction you specify), and creates a new via, dynamically attached to the bond finger. If a bond finger has a cline attached, the tool considers it as already escaped from the die, and skips it. It looks for the next bond finger with no escape clines. When it reaches the last bond finger on the guide path, a message indicates that there are no more bond fingers. The tool exits the *Add Via* mode and reverts to the main pop-up menu.
Lock Via and Stretch Escape	Choose this option when the via is at the desired location, but the tool does not find the preferred escape path. The via locks itself at the current cursor position, and the tool dynamically creates the new escape to pass through the moving cursor. Move the cursor until you are satisfied, and then click to advance to the next bond finger. When you are in this mode, a check mark appears next to the menu option. If you choose *Lock Via and Stretch Escape* again, the tool unlocks the via, the check mark disappears from the menu, and the via resumes tracking the cursor.
Undo Via and Finish	Choose this option to delete the dynamic via and the two clines, thus restoring the bond finger to its original state. The tool restores the main pop-up menu.

Add Via and Finish (dbl-click)	Choose this option (or double-click the mouse in the design) when the dynamic via and escape clines are positioned at the specified location. The tool drops the via and its clines into position. The main pop-up menu is available.
Options North, South, East, West	This option is available only when you drag a via. At that time, two directions are available: either *North* and *South* or *East* and *West*. The selected direction determines which bond finger the tool should select next after you drop the current via. For example, if you are on the north side of the die, the options of *East* and *West* are available. If you choose *East*, the tool selects the next available bond finger to the right of the current bond finger. This option is equivalent to clicking the corresponding buttons in the *Options* panel.

Related Topics

- wire bond escape
- Wirebond Escape Command: Options Panel

Escaping from a Wire Bond Die

Follow these steps to escape from a wire bond die:

1. Choose *Route – Wire Bond – Uprev* to update a pre-Release 16.0 design.
   The design uprev associates the bond fingers to their guide paths.
2. Choose *Route – Wire Bond Escape Generator* from the menu bar.
   Parameters appear in the *Options* panel. The tool also automatically sets the Find Filter to *Vias*, *Fingers*, and *Lines*.
3. Modify the parameters as necessary.
4. Turn on *Wb_Guide_Line* in the *Display* group of the Color and Visibility dialog box by choosing *Display – Color/Visibility* (`color` command).
5. Click on the guide path to select the outer tier of bond fingers.
6. Right-click in the pop-up menu and choose *Generate Escapes*.
   The display updates with the auto-generated escape pattern.
7. Right-click and choose *Deselect Successfully-Escaped Bond Fingers/Via.*
   This action leaves only the failed bond fingers selected.
8. Right-click and choose *Delete Escapes* to delete the failed escapes.
9. Turn off *Wb_Guide_Line* in the *Display* group of the Color and Visibility dialog box.
10. Select a bond finger on the outer tier that has not escaped from the die, or select a bond finger on the inner tier, as near as possible to the center of the die.
11. Right-click and choose *Add Via To Bond Finger*.
    A via appears and dynamically tracks with the cursor. Also tracking are two clines: one between the bond finger and the via, and a second cline between the via and the escape perimeter. The pop-up menu changes when you are in this mode.
12. Click to drop the via at the specified location or right-click and choose *Add Via and Advance*.
    Of the two adjacent bond fingers, the tool, by default, selects the one farther from the die center and creates a new via, which tracks the cursor. Repeat this step as needed until the last bond finger on that side is complete.
13. If vias or bond fingers are not escaping from the specified side of the die (*North*, *South*, *East*, or *West*), then select those vias or bond fingers, right-click in the pop-up menu, and choose *Assign Escape Direction*. Then choose the appropriate direction in the Die Escape: Direction Escape dialog box and click *Assign Escape Direction.*
14. Repeat steps 10 through 13 until all vias are placed for each side of the die.

15. When you complete the die escape task, right-click and choose *Done* from the pop-up menu.
16. Continue with other routing tasks.

Related Topics

- wire bond escape
- Wirebond Escape Command: Options Panel
- Wire Bond Escape: Main Pop-up Menu

wirebond export

The `wirebond export` command lets you export a wire bond pattern from a design to an XML-based file. Then you can import the wire bond pattern to a new design (see the wirebond import command).

For additional information on flows and wire bond tools, see the *Routing User Guide* in your documentation set. For information on design tasks, see Advanced Selection Filtering Dialog Box.

⚠ Bond wires are not exported.

Related Topics

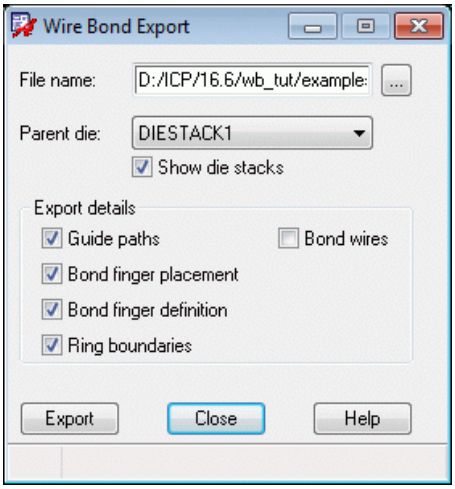
- Exporting Parameters from a Design
- XML-Based Settings File

Wire Bond Export Dialog Box

Access Using

- *Menu Path: Route – Wire Bond – Export*

The Wire Bond Export dialog box appears when you run the `wirebond export` command.



File name	Specifies the name of the wire bond configuration file (`.wbt`) that you are creating. Browse to a different directory using the standard file browser.
Parent die	Specifies the die or die stack from which you are exporting wire bond information. the default setting is the first die or die stack listed alphabetically. In SiP Layout, the list is controlled by the *Show die stacks* check box described next.
Show die stacks	Specifies if the *Parent die* field should list die stacks or dies. If selected, the *Parent die* field lists the die stacks. This field is available only in SiP Layout.
Export details	Lets you choose the information for export.
Guide paths	Select to export the guide path used by the selected group. The default setting is enabled.

Bond finger placement	Select to export the bond finger (x, y location, rotation, and finger label) for bond fingers in the selected groups. The default setting is enabled.
Bond finger definition	Select to export information about the bond fingers, such as padstack name, shape, height, and width.The default setting is enabled.
Ring boundaries	Select to export the boundaries for power and ground rings linked to the groups that you are exporting. If there are no nets, the tool creates them. It also records the net assignments, which are reused during import if the same nets exist in the new design.The default setting is enabled.
Bond wires	Select this box to export bond wire information relating to the end point location, profile, the stack level for wire starts and ends, and the diameter of the wire. The stack level is the offset in the die stack of the die or interposer that the bond wire connects to at the end. ⚠ For wire bond configuration files generated prior to 16.6 QIR 4, the stack level for wire starts and ends will not be available. You need to regenerate the files to include the information. Once regenerated, the files cannot be used any earlier releases.

Related Topics

- XML-Based Settings File

Exporting Parameters from a Design

Perform the following steps to export wirebond parameters from a design:

1. Open the design from which you want to export the wire bond pattern.
2. Run the `wirebond export` command.
   The Wire Bond Export dialog box appears.
3. Specify the wire bond configuration file (`.wbt`) file name and location.
4. Check the information in the dialog box for export.
   You can export a subset of the overall wire bond pattern. Click *OK*.
   The tool exports the wire bond pattern to the wire bond configuration file (`.wbt`). Any errors or warnings that occur are noted in the `wirebond.log` file.

Related Topics

- wirebond export

XML-Based Settings File

The following shows only a very small sample, but illustrates all of the proposed constructs that can be in the configuration file.

```
<?xml version="1.0" encoding="ISO-8859-1" ?>

<!DOCTYPE die SYSTEM "cdnwb.dtd">

<die>

  <die_name>DIESTACK1</die_name>

  <die_height>10000 UM</die_height>

  <die_width>10000 UM</die_width>

  <finger_def>

    <pad_name>BF1</pad_name>

    <shape>OBLONG_X</shape>

    <height>150 UM</height>

    <width>750 UM</width>

    </finger_def>

  <finger_def>

    <pad_name>WB_TACKPOINT</pad_name>

    <shape>CIRCLE</shape>

    <height>0.01 UM</height>
```

```
    <width>0.01 UM</width>

    </finger_def>

  <finger>

    <label>BF2</label>

    <loc_x>7260.55 UM</loc_x>

    <loc_y>10152.6 UM</loc_y>

    <angle>62.737</angle>

    <pattern>On Path</pattern>

    <alignment>Aligned with Wire</alignment>

    <snap>Finger Origin</snap>

    <equal_wire_length>2750 UM</equal_wire_length>

    <pad_name>BF1</pad_name>

    <merge_group>1</merge_group>

    </finger>

<path>

    <arc>

      <x1>-9132.68 UM</x1>

      <y1>9432.68 UM</y1>
```

```
      <x2>9332.68 UM</x2>

      <y2>9432.68 UM</y2>

      <cx>100 UM</cx>

      <cy>-13800 UM</cy>

      <rad>25000 UM</rad>

      <dir>Clockwise</dir>

      </arc>

    </path>

<ring>

    <net>VSS</net>

    <boundary>

      <line>

        <x1>6250 UM</x1>

        <y1>5250 UM</y1>

        <x2>6250 UM</x2>

        <y2>-5250 UM</y2>

        </line>

      <arc>
```

```
      <x1>6250 UM</x1>

      <y1>-5250 UM</y1>

      <x2>5250 UM</x2>

      <y2>-6250 UM</y2>

      <cx>5250 UM</cx>

      <cy>-5250 UM</cy>

      <rad>1000 UM</rad>

      <dir>Clockwise</dir>

      </arc>

    <line>

      <x1>5250 UM</x1>

      <y1>-6250 UM</y1>

      <x2>-5250 UM</x2>

      <y2>-6250 UM</y2>

      </line>

  </boundary>

  <void>

    <line>
```

```
        <x1>5750 UM</x1>

        <y1>5250 UM</y1>

        <x2>5750 UM</x2>

        <y2>-5250 UM</y2>

        </line>

</ring>

<rad>2000 UM</rad>

        <dir>Clockwise</dir>

        </arc>

      <line>

        <x1>-7250 UM</x1>

        <y1>-5250 UM</y1>

        <x2>-7250 UM</x2>

        <y2>5250 UM</y2>

        </line>

      <arc>

        <x1>-7250 UM</x1>

        <y1>5250 UM</y1>
```

```
        <x2>-5250 UM</x2>
</boundary>
    <void>
      <line>
        <x1>6750 UM</x1>
        <y1>5250 UM</y1>
        <x2>6750 UM</x2>
        <y2>-5250 UM</y2>
        </line>
      <arc>
        <x1>6750 UM</x1>
        <y1>-5250 UM</y1>
        <x2>5250 UM</x2>
        <y2>-6750 UM</y2>
        <cx>5250 UM</cx>
        <cy>-5250 UM</cy>
        <rad>1500 UM</rad>
        <dir>Clockwise</dir>
```

```
      </arc>
    <line>
      <x1>5250 UM</x1>
      <y1>-6750 UM</y1>
      <x2>-5250 UM</x2>
      <y2>-6750 UM</y2>
      </line>
<line>
      <x1>-5250 UM</x1>
      <y1>6750 UM</y1>
      <x2>5250 UM</x2>
      <y2>6750 UM</y2>
      </line>
    <arc>
      <x1>5250 UM</x1>
      <y1>6750 UM</y1>
      <x2>6750 UM</x2>
      <y2>5250 UM</y2>
```

```
      <cx>5250 UM</cx>

      <cy>5250 UM</cy>

      <rad>1500 UM</rad>

      <dir>Clockwise</dir>

      </arc>

    </void>

  </ring>

<wire>

  <profile>PROFILE1</profile>

  <diameter>25.4 UM</diameter>

  <wire_start>

    <loc_x>4354.93 UM</loc_x>

    <loc_y>4655.27 UM</loc_y>

    <stack_level>1</stack_level>

    </wire_start>

  <wire_end>

    <loc_x>5218.19 UM</loc_x>

    <loc_y>8300 UM</loc_y>
```

```
    </wire_end>

  </wire>

<wire>

  <profile>PROFILE1</profile>

  <diameter>25.4 UM</diameter>

  <wire_start>

    <loc_x>4204.76 UM</loc_x>

    <loc_y>4805.44 UM</loc_y>

    <stack_level>1</stack_level>

    </wire_start>

  <wire_end>

    <loc_x>5000.12 UM</loc_x>

    <loc_y>8300.01 UM</loc_y>

    </wire_end>

  </wire>

</die>
```

Related Topics

- wirebond export
- Wire Bond Export Dialog Box

wirebond import

The `wirebond import` command lets you import the archived parameters of a wire bond pattern from a previous design into your current design. You seed the new design with the basics of a known pattern, which you can then customize or rework to fit the new design.

⚠ For wire bond configuration files generated prior to 16.6 QIR 4, the stack level for wire starts and ends will not be available. The stack level is the offset in the die stack of the die or interposer that the bond wire connects to at the end. You need to regenerate the wire bond configuration files (`.wbt`) to include the information. Once regenerated, the files cannot be used any earlier releases.

Related Topics

- Importing Setting from a Pre-Release 16.3 XML File
- Importing a Wire Bond Pattern from Another Design

Wire Bond Import Dialog Box

Access Using

- *Menu Path: Route – Wire Bond – Import*

The Wire Bond Import dialog box appears when you run the `wirebond import` command.

File name	Specifies the name of the wire bond configuration file (XML-based) that you load. Using the … button allows you to browse to the file using the standard file browser.
Parent die	Specifies the die or die stack around which the tool should place the imported items. The default setting is the first die or die stack listed alphabetically. In SiP Layout, the list is controlled by the *Show die stack* check box described next.
Show die stacks	Specifies if the *Parent Die* field should list die stacks or dies. If selected, the Parent die field list the die stacks. This field is available only in SiP Layout.
Import data	
Guide paths	If you check this box, the tool imports the guide path information. The default setting is enabled.
Bond finger placement	If you check this box, the tool imports the bond finger placement (x, y location and rotation) and places fingers at those locations. The default setting is enabled.
Bond finger definition	If you check this box, the tool creates any padstack definitions necessary in the new design, based on the information in the configuration file. The default setting is enabled.
Padstack to use	Specifies the padstack definition to use for any fingers in the XML file which do not have a bond finger padstack listed or for which the padstack referenced is undefined in both the database and XML file itself. This field applies to all bond fingers not connected to rings.
Ring boundaries	If you check this box, the tool imports boundaries for power and ground rings specified in the wire bond configuration files. If there are no existing nets, the tool creates them. These rings are connected to the appropriate voltage nets only if the nets already exist in the design. The default setting is enabled.

Bond wires	If you select this box, checks are done for dangling wires in cases where the pins are not in the same place as in the original die. These dangling wires are not imported.
Placement details	
Use values in file as absolute	Check this box to use the values in the file as absolute. This means that the x/y value in the file is the one that is used when you import the settings. If you do not check the box, the values are relative to the origin point of the die or the die-stack to which you are importing the values. For example, if your die is placed at 500,500, and you have a bond finger at (800, 800) in the file you are importing, an absolute value places the bond finger at 800,800, while a relative value places it at (800+500, 800+500).
Apply mirroring	Check this box to apply mirroring. This mirrors all coordinates around the placement origin or center when they are imported. For example, a bond finger that was on the left will now be on the right. Standard mirror behavior.
Rotation	Choose a value from the drop-down list to rotate the design items upon import. This rotation value applies to all items when they are imported. Similar to the mirroring option, rotation is applied around the origin or center of the imported data. For example, if you rotate 90 degrees, a bond finger on the north side is placed on the west side.

Related Topics

- Importing a Wire Bond Pattern from Another Design

Importing Setting from a Pre-Release 16.3 XML File

You can import (wirebond import) an existing bond shell, guide paths, and padstack definitions from a pre-Release 16.3 XML file. All group names are discarded and all the group-level properties are passed to individual wire bonds. The design then follows the *groupless* model.

Group characteristics propagate to the member wires and fingers. Any customization that you performed, for example, a manual override of a wire's profile in contrast to the wire's group profile, keeps the customized values.

For additional information on flows and wire bond tools, see the *Allegro User Guide: Routing the Design* in your documentation set. For information on design tasks, see Advanced Selection Filtering Dialog Box.

⚠ In 16.3 and earlier versions, bond wires are not exported in the `.xml` file with the other archived parameters. Once you import the settings into your design, use the *Wirebond – Select – Reconnect* command to add the bond wires. See Attaching Multiple Wire Bonds to One Bond Finger.

Related Topics

- wirebond import

Importing a Wire Bond Pattern from Another Design

Be sure that your die is placed in the current design and that you have already exported parameters from a design.

1. Open the design to which you are importing parameters.
2. Choose *Route – Wire Bond – Import* (`wirebond import`) from the menu.
   The Wire Bond Import dialog box appears.
   If you are importing parameters into the same design for replication of the pattern to other sides of the same die, configure the new die sides as required. If you are importing parameters into a new design, be sure that you check all four sides of the die.

   ⚠ If the XML file has merge finger groups defined using `merge_group`, the fingers must be on the same net, and must be within 1.5x finger separation to be merged.

3. Complete the parameters and click *Import*.
   The tool imports the elements into the active drawing, and scales them based on the current die's size and aspect ratio versus the same parameters of the die on which the archived pattern was based.
4. Click *Close* to dismiss the dialog box.

Related Topics

- wirebond import
- Wire Bond Import Dialog Box

wirebond lock

The `wirebond lock` command locks all bond wires and fingers to the connected die symbol(s) to restore the standard editing behavior.

By default, bond wires and bond fingers are created linked to the die symbol with a parent and child relationship between the bond pads or fingers and bond wires with the symbol they are connected. The parent and child relationship or locking allows editing commands such as *Move*, *Delete*, *Rotate* to treat the wires and fingers as part of the die symbol, acting on them all as a single unit. For example, if you move a die symbol while locked, the bond wires and bond fingers connected to the die symbol are also moved. Similarly, deleting or unplacing the die symbol rips up the bond items.

You normally use this command after having created an unlock condition using `wirebond unlock`.

Related Topics

- wirebond unlock

Running the Wirebond Lock Command

To lock all bond wires and fingers, follow these steps:

1. In the command window, enter `wirebond lock`.
   A conformation window appears explaining the feature and asking if you want to continue.
2. Click *Yes* to lock the die symbols and their connected wires and fingers in the open design.

wirebond manage guide paths

The `wirebond manage guide paths` command lets you add and edit bond finger guide paths (lines) which the tool follows when placing bond fingers. The guide path is where the connection point of the bond finger (usually in the center of the padstack) is placed when you add bond fingers.

Once you select a wire bond die or discrete component, you can establish a set of guide paths around that component relative to the component's placed extents and pin arrangement. When you select a bond finger guide path, you can manipulate the path while seeing the affects on the pattern of fingers at the same time.

Before wire bonding the design, using this command, you can use the *Populate with Fingers* option to select a guide path and populate it with as many unwired bond fingers as possible using a specified padstack and spacing. Because no bond wires are created by this command, the exact placement and spacing of bond fingers is based only on the supplied bond finger-to-bond finger spacing value. As a result, the tool may place fewer than the actual number of wired bond fingers that can fit on this path in the design.

⚠ In the SI tools, only the `wirebond select` command provides the *Populate with Fingers* option.

Related Topics

- Wirebond Mange Guide Paths - Redistribute Fingers Command: Options Panel
- Wirebond Manage Guide Paths Procedures
- wirebond select

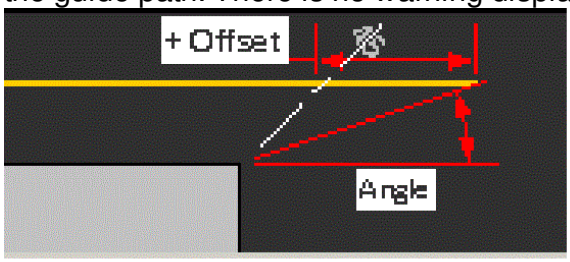
Wirebond Manage Guide Paths - Add Guide Command: Options Panel

Access Using

- *Menu Path: Route – Wire Bond – Add/Edit Guide Paths*

The following parameters appear in the *Options* panel when you run the `wirebond manage guide paths` command and choose the *Add Guide*. You can also use the `wirebond select` command to add and edit a guide path.

Die Side	Specifies the side of the die on which to create the guide path. The default setting is *North*.
Path Characteristics	
Distance	Specifies the distance from either the die edge or the corner die pad at which the tool should place the guide. The default setting is the value of the finger-to-component edge spacing rule currently specified in the Global Wire Bond Constraints dialog box.
from edge of die	If you choose this option, the tool measures the distance from the edge of the currently selected component. This is the default setting.
from nearest die pad	If you choose this option, the tool measures the distance from the center of the outermost die pad, orthogonal to the die edge. This makes the distance value better approximate the actual wire bond lengths used.
Add to die group	Check this box to link the new guide path to the die component. As a result, the guide path moves with the die during a move command.
Radius	Specifies the radius of the new arc path segment. If you check this box, you can enter either an arc radius or 0.0 to specify a straight line. The default setting is 0.0 and creates a straight guide path.

Path vertices defined by	*W*ith this option, you can specify a distance relative to the 45-degree line or angle from the die boundary when creating either linear or arc guide paths (see graphic below). If you have space constrictions, this helps in placement of bond fingers on guide paths. You can specify a negative (inside the 45-degree line) or positive (outside the 45-degree line) distance. The tool applies the distance to both ends of the guide path. There is no warning displayed about crossing guide paths. 
angle	If you click *angle*, you can specify the distance for either a linear or arc guide path by choosing a value in degrees, for example, 52 degrees. The default setting for this field is 46 degrees (prevents the guide paths on all sides from touching).
+/- offset	If you click *offset*, you can specify the distance for only a linear guide path by entering a value in the text box, for example, 500 UM, beyond the 45-degree line. The default setting for this field is -1 UM (prevents the guide paths on all sides from touching). The default setting for linear paths is *Offset*.
Create guide path(s)	Saves the guide in the database.
Done adding guides	Click this button when you are finished adding guide paths.

Related Topics

- Wirebond Manage Guide Paths Procedures

Wirebond Manage Guide Paths - Redistribute Fingers Command: Options Panel

The following parameters appear in the *Options* panel when you run the `wirebond manage guide paths` command and choose the *Redistribute Fingers* menu item.

Path Order	Specifies the order in which to process the pin or finger on the guide path. Options are: • *Inner to Outer* – the tool adds the first finger on the innermost guide path. • *Outer to Inner* – the tool first adds the finger on the outermost guide path. For example, if you have three guide paths, and you choose *Inner to Outer*, the tool places the first finger on the innermost path, the second finger on the next path, and the third finger on the outer guide path. The tool then starts the cycle over again. If you choose *Outer to Inner* for the previous example, the tool reverses the order of placement.
Pin Order	Determines which pin or finger to process on the guide path first. Options are: • *Clockwise – the tool starts with the pin in the set that is farthest away in a clockwise direction, for example, on the North side, this is the pin on the extreme left.* • *Counter-clockwise – the tool starts with the pin in the set that is closest in a counter-clockwise direction, for example, on the North side, this is the pin on the extreme right.* • *Center– simulates the normal bonding order for the tool. This means that the tool works out from the middle of the set of pins in both directions.* The default setting is *Clockwise*.

Related Topics

- wirebond manage guide paths

Wirebond Manage Guide Paths Procedures

You can use the `wirebond manage guide paths` command to perform the following procedures but it is generally easier to use the `wirebond select` command, especially if you are performing multiple tasks. The procedures are described under `wirebond select`command. See:

- Adding a Bond Finger Guide Path
- Editing or Copying the Bond Finger Guide Path
- Moving the Bond Finger Guide Path
- Redistributing Bond Fingers Across Multiple Bond Finger Guide Paths

Related Topics

- wirebond manage guide paths
- Wirebond Manage Guide Paths - Add Guide Command: Options Panel

wirebond purge groups

The `wirebond purge groups` command lets you remove some or all unused groups left in your design database when you remove or replace bond fingers. These groups are not associated with any bond fingers, but still remain listed as groups.

Related Topics

- Purging Unused Wire Bond Groups from your Design

Purge Unused Wire Bond Groups Dialog Box

Access Using

- *Menu Path: Route – Wire Bond – Purge Unused Groups*

The Purge Unused Wire Bond Groups dialog box lists all unused wire bond groups in the current design.

Delete	Lists all unused wire bond groups in the current design that you are going to delete.
Save	Lists the unused wire bond groups that you elect to keep in the design database
Save all ->	Moves the entire list of unused wire bond groups to the *Save* pane.
<- Delete all	Moves the entire list of unused wire bond groups to the *Delete* pane.
OK	Removes all the wirebond groups from your design that are listed in the *Delete* pane, closes the dialog box, and returns the tool to an idle state
Cancel	Closes the dialog box without changes

Purging Unused Wire Bond Groups from your Design

Follow these steps to purge unused wire bond groups from the layout:

1. Run the `wirebond purge groups` command at the command window prompt.
   If there are unused wire bond groups in your design, the Purge Unused Wire Bond Groups dialog box appears. It lists all the unused wire bond groups in the left pane. If the database contains no unused wire bond groups, that message appears in the command window and the tool remains in the idle state.
2. Determine which, if any, unused wire bond groups that you want to retain in your design database by:
   - Clicking on individual group names to move them into the *Save* panel
   - Clicking *Save all ->* to retain all unused groups
3. To remove all unused nets in the design, click *OK*.
4. To see the results of the purge operation, run the `viewlog` command to display the log file, `purgeUnusedWB Groups.log`.
   If the system is unable to purge one or more unused wire bond groups, descriptive error messages are displayed at the command window of the user interface.

Related Topics

- wirebond purge groups

wirebond reports

The `wirebond reports` command lets you extract wire bond and bonding pattern information from your design and create a report. You can customize the report to display only the information you want. You can configure the number and type of columns and sort the report by any data such as net name, pin name, finger name, and so on. You can configure the order of the columns from left to right. The tool displays the report in a tab-delimited format for import into a spreadsheet.

Related Topics

- Generating a Wire Bond Report

Wire Bond Reporting Dialog Box

Access Using

- *Menu Path: Reports – Wire Bond Report*

File name	Specifies the name of the report file if you choose to save your report to disk. Click... to browse to a directory and file name. You must have write access to the specified directory. The default file name is `wirebond_report.txt`
File type	Click the button for the format in which you want the report. Choices are: RPT, TXT. and HTML.
Sort column	Provides a pull-down list by which you can specify a data field for sorting the report entries. You can sort by all the fields listed in the *Report Fields* below, except for *Index*. You do not have to select the value of the *Sort* field for display in the report. The default setting is *Net Name.*
Show wire length totals	Includes a summary table of total wire lengths broken out by material and wire diameter. Not selected by default.
Report fields	Configuration files are text files listing the default report name, sort field, and an (ordered) list of columns to include in the report.
Load report config	Lets you load your report configuration file.
Save report config	Lets you save a report configuration file, based on your specifications.

	This table lists all the fields, in sorted order, that you can use for export. Therefore, the first row represents the first column of the report, and so on. To remove a column, select the entry and choose the <*Remove*> option at the top of the drop-down list. To add an entry, select the row in the table representing the column before which you will insert the new entry, and then choose the new column name from the drop-down list. If that field already exists in the list, it is moved from its current location to the new one. Choose *Index* from the list to specify numbering for each row. Options for sorting include (Items marked with an asterisk (*) are the default settings: ◦ Index ◦ Net Name * ◦ Pin Number * ◦ Pin Location ◦ Pin Wire Count ◦ End Pin Number ◦ End Pin Location ◦ Finger Name * ◦ Finger Padstack ◦ Finger Location ◦ Finger Layer ◦ Finger Wire Count ◦ BGA Ball Number ◦ BGA Ball Location ◦ 3D Wire Length ◦ 2D Wire Length * ◦ Wire Diameter ◦ Wire Type

Report fields (cont)	◦ Wire Profile ◦ Wire Start Location ◦ Wire Start Layer ◦ Wire End Location ◦ Wire End Layer ◦ Wire Material ◦ Wire Start Height Relative to Substrate ◦ Wire End Height Relative to Substrate ◦ Wire Die Side ◦ Wire Angle, Relative to Die Side * ◦ Finger Angle, Relative to Die Side ◦ Finger Angle, Relative to Wire Angle ⚠ The report contains values of --- whenever a column does not have a value for the specific row. For example, *Finger Padstack* would not have a value for a die-to-die bond wire. This report does not include unwired pins by default. To include unwired pins, set the WIREBOND_REPORT_INCLUDE_PINS variable in the IC Packaging section of the User Preferences dialog box.
Default columns	Click to restore the selected columns to the default settings and their default order.
Add all columns	Lets you add all the report fields (columns of the formatted report).
Remove all columns	Lets you remove all report fields (columns of the formatted report).
Write report	Check this box to save the report to disk with the specified file name. This setting is enabled by default.
View report	Check this box to display the report. This setting is enabled by default.

Report	Click to generate the report per your specifications. The tool displays the report if you select *View Report* and saves the file to disk if you select *Write Report*.
Close	Click to close the Wire Bond Reporting dialog box and return to the Design window.
Help	Click to display help for this command.

Generating a Wire Bond Report

Perform the following steps to generate a wirebond report:

1. Run the `wirebond reports` command.
   Items in the Find Filter become available for selection.
2. Configure settings for the report. Choose a name, the fields, and the order for which you want to display each field. Also, select a field by which to sort.
3. Choose to view the report or save it to disk, or both.
4. To focus your report on specified items in the design, check the appropriate box in the Find Filter and click on the element in the Design Window.
   If you do not select any items, the report lists all items in the design.

   ⚠ You can filter by Pins if you set the environment variable `WIREBOND _REPORT_INCLUDE_PINS` in the *IC Packaging* section of User Preferences.

5. Click *Report* to create the report.
6. When finished generating reports, click *Close* to return to the Design Window.

Related Topics

- wirebond reports

wirebond select

The `wirebond select` command is the central command of the wire bonding tools. Based on the action items that you select in the Design Window and the items selected in the pop-up menus, the `wirebond select` command lets you perform all wire bond tasks except for moving the endpoint of a bond wire within the bond finger area (`wirebond tack point`), importing configuration settings (`wirebond import`), and exporting configuration settings (`wirebond export`).

By default, the tool does not let you add multiple wire bonds to a pin. To change this behavior, set the `wirebond_multwire_pins` environment variable and connect multiple wire bonds to a single pin. You can set this variable by choosing *Setup – Preferences* (`enved` command), then clicking the *IC_packaging* category and then *Wirebond*. For additional information on flows and wire bond tools, see the *Routing User Guide i*n your documentation set.

Access Using

- *Menu Path: Route – Wire Bond – Select*
- Toolbar Icon: 

Related Topics

- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks
- wirebond tack point
- wirebond import
- wirebond export

Dialog Boxes and Options Panels for the Wirebond Select Command

Advanced Selection Filtering Dialog Box	Advanced Selection Filtering
Wirebond Select Command - Perform Auto Bonding Command: Options Panel	Wirebond Select Command - Populate with Fingers Command: Options Panel
Wirebond Select Command - Change Characteristics Command: Options Panel	Wirebond Select Command - Create Ring Command: Options Panel
Wirebond Select Command - Edit Routing Stubs: Options Panel	Wirebond Select Command - Add Jumper Command - Options Panel

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Advanced Selection Filtering Dialog Box

If you checked the *Use advanced selection filtering* box in the Wire Bond Settings dialog box (`wirebond settings` command), and then run the `wirebond select` command to add bond fingers, the Advanced Selection Filtering dialog box appears.

Related Topics

- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Advanced Selection Filtering

The *Advanced Selection Filtering* option lets you filter nets or pins, or both, when you run the following commands:

- auto assign pinuse
- auto assign net
- assign route layer
- assign plating layer
- wirebond select
- wirebond add

Once you select the *Use advanced selection filtering* option and then select the nets or pins in the design, the Advanced Selection Filtering dialog box appears. The tree view displays top-level items (the nets) that you can click on to see the pins associated with them.

By default, the *Filter* field displays an asterisk (*) which means that the list displays all the selected nets. You can modify this field to display a list that is easier for you to manage.

Clicking the net name automatically selects or deselects all the associated pins.

Wirebond Select Command - Perform Auto Bonding Command: Options Panel

These settings appear in the *Options* panel when you run the `wirebond select` command to perform auto bonding. See The following table lists the elements selected in the Find Filter and the corresponding tasks that you can perform with the `wirebond select` command..

Bond wires	
Profile	Specifies the wire bond profile to use for auto bonding. By default, this field is set to the profile specified as default in wire bond settings.
Power/Ground rings	
Create rings	Specifies whether the tool creates power and ground rings. The ground ring is closest to the die. The default setting is unchecked. ⚠ The tool creates rings with rounded corners, using the default setting of the `pring wizard` command.
Dist. to 1st ring	Specifies the distance between the die edge and the first ring boundary. By default, this is 7.5% of the minimum of die width and die height (same as in the `pring wizard` command).
Ring width	Specifies the width of the ring.
Dist. between rings	Specifies the distance between the two rings. By default, this is 7.5% of the minimum of die width and die height (same as in the `pring wizard` command).
Guide paths	
Linear path	Specifies whether the tool creates a linear guide path. If you do not check this box, the bond finger guide path is an arc. By default, this box is not checked.
Row bond	Check this box to enable row bonding. The tool calculates the pin rows on each die side and creates the same number of guide paths as the number of pin rows per side. It does not consider the value you enter for the number of guide paths in the *Options* panel. The die or diestack is auto-bonded with the row closest to the die edge being bonded to the closest guide path to the die. Pin rows are determined by pin (x,y) location. If you enable row bonding, the *Paths per die side* field is disabled. By default, row bonding is *Off*.
Paths per die side	Specifies the number of paths that the tool creates per die side.

Dist. to 1st path	Specifies the distance between the first guide path on a side to the die edge if you do not create rings, or to the outer ring boundary if you create rings. By default, this setting equals the value of the *Dist. to 1st ring* field.
Dist. between paths	Specifies the distance between guide paths on one side. By default, this setting equals the value of the *Dist. between rings* field.
Auto bond	Starts the auto bond process. ⚠ The tool uses the default bond finger padstack (`wirebond settings` command) for the bond fingers.
Done auto bond	Completes the process and exits the command.

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wirebond Select Command - Populate with Fingers Command: Options Panel

Spacing	Specifies the bond finger-to-bond finger spacing that the tool should use. This value defaults to the general bond finger-to-bond finger spacing value in the DRC constraints. You can override this value to account for same net bond fingers, routing channels, and so on. If you enter a value less than the DRC constraint value, DRCs result.
Bond Finger	Specifies the bond finger padstack that the tool uses for adding fingers to the selected paths. Choose from the drop-down list or click *Add* to create a new padstack. This value defaults to the default finger padstack specified in the wire bond settings.
Wire Profile	Specifies the wire profile that you want to use when you add bond wires to the bond fingers you create with this command.
Create bond fingers	Click this button to save the bond fingers to the design.
Number of bond fingers that will be created given current settings	A status indicator that indicates the total number of fingers created, using the current settings.
Number of new DRCs based on current settings	A status indicator showing the number of new DRCs resulting from the bond finger creation, using these settings.

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wirebond Select Command - Change Characteristics Command: Options Panel

You can change characteristics for a set of bond fingers or change a wire profile using the *Options* panel. You can see the resulting changes in the bond pattern in the Design Window as you make the changes.

Pattern	
Style	Specifies the wire bond pattern. Choices are: • *Orthogonal – Specifies a pattern of* 0-degree bond wire angles and bond pads. • *Equal Wire Length* – Specifies that wire bond angles are variable, but all wire bond lengths are equal. • *On-Path* – Specifies that all wire bonds stay on the bond finger guide path. • *Free Placement* – Does not snap to any path. With this option, the fingers are never automatically pushed or shoved due to bubbling as they have no associated path of movement. This field determines the placement rules applied based on the path that these elements follow. The default setting is *On-Path unless there are no guide paths; then the default setting is Equal Wire Length*. This field is read-only if you select *Bond wires* in the Find Filter.
Length	Specifies the wire length. This field is available only if you select *Equal Wire Length* as a style. ⚠ For non-wired bond fingers, this length is approximated for the placement of non-wired bond fingers since there is no measurable length of wire from pin to bond finger. For non-wired bond fingers that have the *On Path* pattern but have no paths because they were removed after the group was created, the placement is approximate and not based on the value in the *Length* field since there is no path to snap to.
Wire	

Profile	Specifies the wire bond profile (including diameter) to associate with this wire. A profile determines the approximate 3D path that the wire bond follows. To specify the name of a new profile, enter the name in this field and then use the *Route – Wirebond – Settings – View/Edit wire profiles* to create the profile in the profile editor. Then you can select the profile from the pull-down menu. This field is read-only if no wire has been selected.
Finger	
Pad	Lists the available padstacks that you select for the bond fingers. The default padstack is specified in the Wire Bond Settings dialog box (see the wirebond settings command). This field is read-only if no finger has been selected.
Add	Displays the Finger Padstack dialog box which allows you to define a new finger padstack.
Align	Specifies the orientation to use when placing bond fingers. Choices are: • *Aligned with Wire* • *Orthogonal to Die Side* • *Orthogonal to Guide* • *Pivoting Ortho to Guide* • *Average Wire Angle* • *Constant Angle* • *Match CW Neighbor* • *Match CCW Neighbor* The default setting is *Aligned with Wire*. Choose the *Pivoting Ortho to Guide* option for oblong or rectangular bond fingers. It lets you avoid the situation when the wire is almost perpendicular to the die side and the bond wire crosses the boundary of the bond finger over the rounded end of the bond finger. As the bond wire angle increases, the bond wire starts to cross the bond finger boundary on one of its straight edges. With this option, the tool pivots the bond finger slightly so that the bond wire crosses over the rounded end. This field is read-only if no finger has been selected.
Angle	Specifies the value for the angle alignment. This field is read-only unless you select *Constant Angle* alignment.

Snap	Controls which point on the bond finger snaps to the guide path. Because bond fingers vary in size, sometimes it is necessary to control how they follow the path so that bond wire lengths are equal and you achieve the smoothest possible pattern. Options are: • *Center of Finger* • *Finger Origin* • *Near End* (the tip of the bond finger closest to the die edge) • *Far End* (the tip of the bond finger farthest from the die edge) • *Nearest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is closest to the die) • *Farthest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is farthest from the die) The default setting is *Finger Origin*. This field is read-only if you have not selected a bond finger.

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wirebond Select Command - Create Ring Command: Options Panel

The *Options* panel displays the following parameters when you use the `wirebond select` (or `wirebond edit`) command with the *Advanced – Create Ring* menu item to create a ring segment from a set of placed bond fingers.

Clearances	
Inside	Specifies the expansion distance to apply to the inside edge of the shape (end of the fingers pointed towards the die edge). The default value is 0 UM.
Outside	Specifies the expansion distance to apply to the outside boundary of the shape (end of the fingers pointed away from the die). The default value is 0 UM.
Ends	Specifies the expansion distance to apply to the two ends of the ring segment when generated. The default value is 0 UM.
Create rectangular shape	Click this box to generate a plain rectangle based on the extents of the convex hull computed. Using this option makes it easier to slot ring segments together, modify, and get a clean pattern. The default value is disabled.
Generate	Creates the new ring section, its net assignment (same as the old bond fingers), and the replacement of old fingers with shape tack connections.

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wirebond Select Command - Edit Routing Stubs: Options Panel

You can also use the `wirebond edit` command to display this *Options* panel.

Routing Stubs	
Outer	Check this box to modify or add a stub from the bond finger in the direction away from the die edge. Uncheck the box to remove stubs in this direction. The default setting is *Off*, which causes the *Length* and *Padstack* fields to be disabled.
Inner	Check this box to modify or add a stub from the bond finger in the direction toward the die edge. Uncheck the box to remove stubs in this direction. The default setting is *Off*, which causes the *Length* and *Padstack* fields to be disabled.
Outer / Inner Length	Specifies the length of the stub, which is measured from the end of the bond finger from which the stub exits. By default, it is twice the width and length of the default bond finger padstack to ensure that there is clearance from the bond finger. The tool always aligns the stubs with the bond finger's rotation to ensure a good routing from the die for routing and manufacturing. The tool always aligns the stubs with the bond finger's rotation to ensure optimal routing from the die for manufacturing and wire bond assembly. If you want an off-angle fanout, you can change it using the *Edit – Vertex* command. If you have multiple length stubs for the selected items, ** is displayed in the *Outer* or *Inner* fields.
Outer /Inner Via Pad	Specifies the padstack to use for the via at the end of the routing stub, if desired. By default, this field is set to <*No Via*>, and no via is added. If you have multiple via pads for the selected items, ** is displayed in the *Via* fields.
DRC Count	Displays the number of additional DRC errors resulting from the current settings.
Update Routing Stubs	Click this button when you have finished changes to the *Options* panel. This commits the changes to the database and returns you to the idle wire bonder state.

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

Wirebond Select Command - Add Jumper Command - Options Panel

Wire	
Allowed multiple wires to finger	Check this box to allow the tool to add multiple bond wires to a bond finger.
Profile	Lets you select the profile for the jumper.

Related Topics

- wirebond select
- Advanced Selection Filtering Dialog Box
- Assign Nets Dialog Box
- Dialog Boxes and Options Panels for the Wirebond Select Command
- Generic Wire Bond Design Tasks
- Element Selection in Find Filter and Corresponding Wire Bond Tasks

wirebond settings

The `wirebond settings` command lets you assign settings for bond fingers, bond wires, and placement. You can:

- View, define, rename, and edit wire profiles (see Wire Bond Profile Editor).
- Define bond finger options such as padstack, wire alignment control when placing bond fingers, and snap point of wires to fingers to guide paths (see Wire Bond Settings Dialog Box).
- Configure for and automatically update bond finger labels (see Bond Finger Labels Dialog Box).
- Set push and shove mode as well as wire bond pattern.
- Allow DRC violations during wire bond manipulations.
- Enable the use of advanced selection filtering when adding bond fingers.
- View and edit (only in *Feasibility* mode) global wire bond constraints (see Global Wire Bond Constraints Dialog Box).
  Cadence recommends that you set constraints using the Constraint Manager

⚠ The command opens a simpler form of the Wire Bond Settings dialog box if run from Allegro X PCB Editor. The form contains only the *Default Finger Options* group of the dialog box.

To access these settings, choose the *Route – Wirebond – Settings* menu command, type `wirebond settings` at the command line, or choose *Settings* from the pop-up menu when you are in another `wirebond` command.

For additional information on flows and wire bonding tools, see the *Wire Bond Use Models* in the *Routing User Guide* in your documentation set.

Related Topics

- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Predefined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Wire Bond Settings Dialog Box

Access Using

- *Menu Path: Route – Wire Bond – Settings*
- Toolbar Icon: 

The Wire Bond Settings dialog box appears when you run the `wirebond settings` command. Use these parameters to set up the default wire bond characteristics and profiles early in the design process. These default settings are then used during the bonding operation. However, you can change these settings during add or move operations.

All the fields in this dialog box are not available in Allegro X PCB Editor. The fields available are for *Default Finfer Options*: *Padstack*, *Alignment*, *Allow cavity placement*, and *Label bond fingers*.

Default Wire Bond Options	
Profile	Specifies the default wire profile used for new bond wires. Initially, the default setting is the first profile defined alphabetically or PROFILE1 if there are no current profiles defined. To add new profiles, click *View/Edit wire profiles*.
View/Edit wire profiles	Displays the Wire Profile Editor (see Wire Bond Profile Editor). To know more about wire profiles, see the *Wire Bonding Toolset*, in *Routing the Design* user guide.
Default Finger Options	
Padstack	Defines the default padstack to use for any bond fingers created in the system. The field initially contains the first padstack name listed alphabetically, which exists on the top substrate layer.

Alignment	Specifies the orientation to use when placing bond fingers. Choices are: • *Aligned with Wire* • *Orthogonal to Die Side* • *Orthogonal to Guide* • *Pivoting Ortho to Guide* • *Average Wire Angle* • *Constant Angle* • *Match CW Neighbor* (bond finger sets its rotation to match that of the clockwise finger beside it.) • *Match CCW Neighbor* (bond finger sets its rotation to match that of the counter-clockwise finger beside it.) The default setting is *Aligned with Wire*. Choose the *Pivoting Ortho to Guide* option for oblong or rectangular bond fingers. It lets you avoid the situation when the wire is almost perpendicular to the die side and the bond wire crosses the boundary of the bond finger over the rounded end of the bond finger. As the bond wire angle increases, the bond wire starts to cross the bond finger boundary on one of its straight edges. With this option, the tool pivots the bond finger slightly so that the bond wire crosses over the rounded end.
Snap point	Controls which point on the bond finger snaps to the guide path. Because bond fingers vary in size, sometimes it is necessary to control how they follow the path so that bond wire lengths are equal and you achieve the smoothest possible pattern. Options are: • *Center of Finger* • *Finger Origin* • *Near End* (the tip of the bond finger closest to the die edge) • *Far End* (the tip of the bond finger farthest from the die edge) • *Nearest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is closest to the die) • *Farthest Point* (computes not the end point along the axis of the wire but instead the exact point on the pad that is farthest from the die) The default setting is *Center of Finger*.

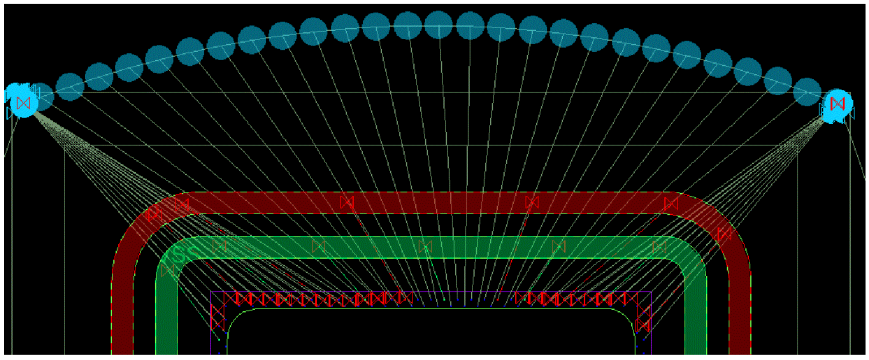
Allow cavity placement	Controls whether or not fingers are allowed to be placed on internal layers exposed by cavities. If disabled (default state) single layer padstacks on internal layers are not listed in the available padstacks pull-down for selection, and if a new finger padstack is defined, that form only allows selection of the top/bottom substrate layers. When the option is enabled, internal padstacks and layers are selectable. This is a global setting and is saved with the design
Label bond fingers	If you check this box, the tool updates the bond finger labels automatically when you add, remove, or manipulate bond fingers. The default setting is disabled.
Configure	Lets you configure the controls for automatic labels in the Bond Finger Labels Dialog Box.
Default Placement Options	
Bubble	Specifies the method for pushing or shoving existing bond fingers and bond wires. Once you choose a method, it is saved as the default with the design even when the current layout session ends and you start a new one. You can choose: • *Shove all* – Pushes and shoves all items necessary based on cursor movements. This is the default setting. • *Shove path* – Pushes and shoves only those items on the same path as the selected items. • *Shove Off* – Does not push or shove anything not selected for movement. It updates the heads-up display showing the state of the minimum and maximum wire length and wire angle values.

Style	Specifies the wire bond pattern for bond fingers. Choices are: • *Orthogonal – Specifies a pattern of* 0-degree bond wire angles and bond pads. • *Equal Wire Length* – Specifies that wire bond angles are variable, but all wire bond lengths are equal. • *On-Path* – Specifies that all wire bonds stay on the bond finger guide path. • *Free Placement* – Does not snap to any path. With this option, the fingers are never automatically pushed or shoved due to bubbling as they have no associated path of movement. This field determines the placement rules applied based on the path that these elements follow. The default setting is *On Path unless there are no guide paths; then the default setting is Equal Wire Length*.
Deflection clearance	Check this box and add a value in the fill-in box so that during the wire bond *Add* or *Move* commands, an X-shaped wire-bond keepout zone is created (the keepout zone is not visible to you). The X is centered on the die or the die stack and the lines of the X pass through the corners of the die or die stack extents. The line thickness of the X is equal to twice the *Deflection Clearance* value that you enter. During cursor movements and final placement, fingers of the wire bonds are not permitted to fall within the thick X lines.
Path snap distance	Controls the snapping capability of the bond fingers to the guide path. If you check the box and specify a value, the tool checks the distance of your cursor from a bond finger guide path. If the cursor distance from the guide path exceeds the specified value, the tool changes the bond finger *Style* from *On Path* to *Free Placement.* If the cursor distance from the guide path is less than the specified value, the bond fingers snap to the guide path.

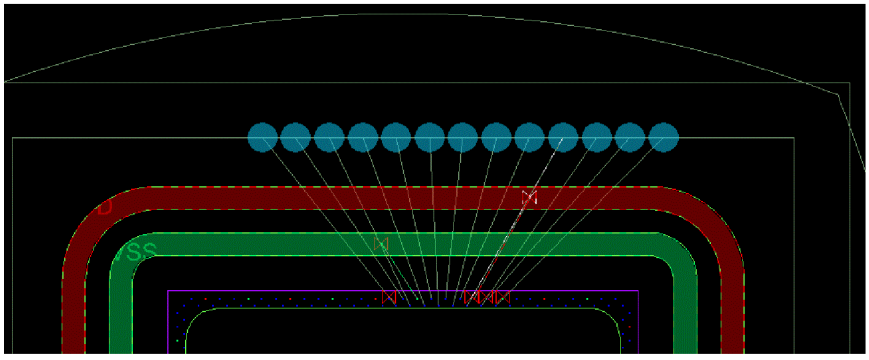
Allow shoved fingers to be placed with DRCs

Check to create a pattern based only on the bond finger-to-finger spacing rule during wire bond manipulations, ignoring all other DRCs such as wire bond spacing, finger-to-ring spacing, and so on. The tool will display the DRCs but will allow you to perform the operations violating these DRCs. You can interactively decide how to fix the DRCs.

For example, while adding wire bonds to a set of pins with the option selected as shown in the following figure, DRCs are displayed but all wire bonds are added.



Without selecting the option, for the same set of pins as shown in the next figure, only those wire bonds that do not violate any DRCs or violate only finger-to-finger DRCs are allowed.



The default setting is enabled.

Use advanced selection filtering	Check this box to have selected items appear in the Advanced Selection Filter dialog box when you are adding bond fingers with the `wirebond select` or `wirebond add` commands. See Advanced Selection Filtering for additional information. The default setting is *Off*.
Pre-defined Settings Groups	
View/Edit settings groups	Displays the Pre-defined Wirebond Settings dialog box.
Feasibility	
Feasibility mode	Click this button to modify and save the wire bond constraint settings in the Global Wire Bond Constraints dialog box while doing feasibility tests. Typically you set these constraints using the Constraint Manager.
View/Edit constraints	Displays the Global Wire Bond Constraints dialog box (see Global Wire Bond Constraints Dialog Box).

Related Topics

- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Bond Finger Labels Dialog Box

The Bond Finger Labels dialog box appears when you click *Configure* in the Wire Bond Settings dialog box. Use it to configure automatic bond finger label generation.

This dialog box complements, but does not replace, the existing capabilities for bond pad label generation. If desired, you can still configure each label in a custom fashion to suit your needs.

> ⚠ To apply bond finger labels to individual dies in SiP Layout, set the `wirebond_sip_label_single_dies` variable in the Wirebond category under lc_packaging in the User Preferences Editor.

To make the labels visible, be sure that you choose *Manufacture – Documentation – Display Pin Text*.

Refdes	Specifies the reference designator of the component or die stack to which the listed label configuration applies.
Numbering	
Use die pin numbers	If checked, the tool uses the attached die pad number as the label assigned to the bond finger. As a result, labels are order-independent. The default setting is *Off*.
Prefix	Specifies the name prefix to apply to every bond finger in this set. The default setting is BF.
Start at	Specifies the number or label with which the tool should begin the numbering. In situations with multiple components, you can configure this to be either a text value or the name of another component or stack. If you choose another element, then the numbers for this pattern begin with the next number available based on those used in the specified stack. This allows for chaining of numbers to ensure that all bond pad labels are unique. The default is no prefix. ⚠ A component or stack can have only a single start at *parent*, and a single component or stack can only be a parent to one other component or stack. This helps prevent duplication and circuitous labeling of fingers.
Increment	Specifies the incremental value to use for each consecutive finger in the pattern. The default setting is 1.
Sorting	

Sort by bond finger location	If you click this button, the tools sorts the bond fingers based on the bond finger placement information. This is the default setting.
Sort by die pin location	If you click this button, the tools sorts the bond fingers based on the die pad positions.
Primary	Specifies the primary sort to apply to the bond fingers. The choices are: *Clockwise* or *Counter-Clockwise*. The tool processes all bond fingers of the pattern at the same time. This means that the search only goes around the circle one time. In the case of multiple guide paths, where two pins or fingers are at the same position in the order, the tool places the outermost element first in the sort. The default setting is *Clockwise*.
Start	Specifies the corner at which to start the numbering pattern. You can choose from any of the four bounding box corners: *Top Left*, *Top Right*, *Bottom Left*, or *Bottom Right*. The default setting is *Top Left.*

Related Topics

- wirebond settings
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Global Wire Bond Constraints Dialog Box

The Global Wire Bond Constraints dialog box appears when you click the *View/Edit constraints* button in the Wire Bond Settings dialog box. The settings in this box are read-only if you did not click the *Feasibility* check box in the Wire Bond Settings dialog box.

The Global Wire Bond Constraints dialog box has two tabs: *Fingers* and *Wires* and a description panel. The tabs contains\ the constraint values for all layers in the design. Typically, you set these constraints using the Constraint Manager.

⚠ By default, during wire bond placement, the tool attempts to satisfy wire-to-wire, finger-to-wire, and finger-to-finger constraint values whether or not these values are currently enabled for online DRC checks. By setting the WIREBOND_IGNORE_DISABLED_CNS environment variable in the User Preferences Editor (`enved` command), the constraint is not used to guide placement if it is not enabled for DRC. If the same net or differential pair bond finger-to-bond finger spacing constraint is disabled, the tool uses the general finger-to-finger spacing value between these types of fingers. Finger-to-finger spacing and wire-to-wire spacing are always used to ensure that placement does not introduce unnecessary electrical shorts.

Fingers Tab

This tab displays all the physical and spacing rules that apply to the bond fingers in the design.

⚠ By default, the tool respects wire profiles when computing legal bond wire placements. As a result if two wires use different profiles, it allows them to cross within a set distance from the die or pin. To avoid the crossing of wires in your design during pushing and shoving of bond fingers (such as when optimizing the design for optical checking), set the `wirebond_ignore_wire_profiles` environment variable in the User Preferences (`enved` command). Setting the `wirebond_ignore_wire_profiles` environment variable causes the tool to treat all wires as part of the same profile for placement only. DRC computations, 3D views and 3D analysis are unaffected by this setting.

⚠ These constraints are read-only, unless you click the *Feasibility* button in the Wire Bond Settings dialog box.

Bond Finger Constraints	

Finger to Finger	Specifies the minimum separation required between two bond fingers on the same substrate layer. To further specify your requirements, check one or more of the boxes: *General*, *Same Net*, or *Diff pair.*
Specialization rules for same net and differential pair finger-to-finger spacing are available in the Constraint Manager tool.	
Finger to	
Wire	Specifies the minimum separation required between a bond finger and a bond wire on the same side of the substrate. This constraint applies to wire bond dies mounted on the either the top or the bottom side of the package substrate. If checked, this rule applies regardless of the wire bond profile used.
Shape	Specifies the minimum separation required between a bond finger and a filled shape on the same substrate layer. This rule does not apply to shapes used for multi-point connection bond fingers.
Via	Specifies the minimum separation required between a bond finger and a via pad that is on the same layer as the bond finger.
Route	Specifies the minimum separation required between a bond finger and a routing wire that is on the same layer as the bond finger.
Pin	Specifies the minimum separation required between a bond finger and a component pin pad defined on the same side of the substrate. This constraint applies only to wire bond dies mounted on the bottom side of the package substrate.
Component	Specifies the minimum separation required between a bond wire and a pin pad defined on the same layer. The pad will normally be a pin on the same component and may be checked either at the start point of the wire (all wire bonds) or at the end of the wire (die-to-die bonds). This is a two-dimensional check and does not consider the wire profile.

Wires Tab

This tab displays items that govern the physical and spacing rules to apply to the bond wires in the design.

⚠ These constraints are read-only, unless you click the *Feasibility* button in the Wire Bond Settings dialog box.

Bond Wire Constraints	
Wire to wire	
Same profile	Specifies the minimum separation required between two bond wires. This constraint applies to bond wires that share the same wire bond profile and therefore are approximately the same height. These wires cannot cross, or a short occurs.
Diff profile	Specifies the minimum separation required between two bond wires. This constraint applies to bond wires that use different wire bond profiles. As a result, there may be sufficient Z-axis clearance between the bond wires in the region near the die. This constraint applies outside the region controlled by the *Cross length* field.
Cross length	Bond wire-to-wire spacing is the minimum separation required between two bond wires. In cases where two bond wires follow different wire profiles, there may be sufficient Z-axis clearance between the bond wires in the region near the die. This constraint defines the region around the die that allows bond wires with different profiles to cross.
Wire end to	
Wire end	Bond wire end-to-end is the minimum separation between the two connection points of two different bond wires. This applies at both ends of the wires. The value that you input is based on the size of the bonding machine's capillary and arm, which must clear existing wires when creating new wires. At the wire end points, the capillary is closest to the substrate or die surface. This is where the widest portion of the capillary of the machine may impact existing bond wires.
Wire to	
Pin	Specifies the minimum separation required between the connection points of two different wire bonds. The value that you input is based on the size of the bonding machine's capillary and arm, which must clear existing wires when creating new wires. At the wire end points, the capillary is closest to the substrate or die surface. This is where the widest portion of the capillary of the machine may impact existing bond wires.
Wire physical properties	
Min Length	Specifies the minimum length allowed for any bond wire. A bond wire’s length in 2D is the sum of the straight line length of all segments of the bond wire.

Max Length	Specifies the maximum length allowed for any bond wire.
Max Angle	Refers to the maximum angle at which a wire may be placed. This angle is measured relative to the side of the die the wire crosses.
Diameter	Specifies the required diameter to use when bonding the die into the package. Use this value when calculating clearance values for bond finger-to-wire and bond wire-to-wire spacing checks.

ⓘ The constraint values specified in the Global Wire Bond Constraints dialog box are displayed as design-level constraints in Constraint Manager. Profile specific constraint values can be defined using Constraint Manager.

Description Panel of the Global Wire Bond Constraints Dialog Box

This area provides a pictorial and textual description of the current rule, what it applies to, and how it is computed. To save space and remove definitions from the display, you can disable this panel. Run the enved command and set the `wirebond_cns_nodescription` variable under the *Wirebond* category of the User Preferences Editor dialog box.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Pre-defined Wirebond Settings Dialog Box

The Pre-defined Wirebond Settings dialog box appears when you click *View-Edit settings groups* in Pre-defined Settings Groups.

Use this dialog box to load choose active groups, add a new group, and define new groups. In the groups you can define bubble, pattern, wires finger, and routing stubs.

Group Definitions Available	
Active Group	Specifies a list of all groups currently defined in the design or in the *Master Definitions* file.
Add	Lets you add a new group name to the list of groups and to the design. This becomes the *Active Group* definition for editing.
Copy	Lets you add a new group name to the list. The group is a copy of the existing active group with the new name that you provide.
Delete	Lets you remove the active group from the design.
Master Definitions	Specifies a list of all the pre-defined choices of groups listed alphabetically and a *Browse* item that allows you to select a *Master Definitions* file that resides outside the TECHPATH. When you choose *Browse*, the tool displays a file browser to allow you to select the definitions file from disk, starting in the current working directory.
Save	Lets you save the current group to a specified XML file.
Definition	Specify the group definitions.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Wire Bond Profile Editor

The Wire Bond Profile Editor appears when you click the *View/Edit wire profiles* button in the Wire Bond Settings dialog box.

Cadence has worked with the wire bonding machine manufacturers to supply default wire profiles, which are known to be compatible with specific bonding machines. These profile sets are included in the *Master Definitions* list. While you are encouraged to use these profiles, any modifications you make to them may not be compatible with the vendors' machines.

For additional information, see *Creating and Modifying Wire Profiles* in the *Routing User Guide*.

Profiles in Use / Available	
Active Profile	Specifies a list of all profile names currently defined in the design or in the *Master Definitions* file.
Add	Lets you add a new wire profile name to the list of profiles and to the design. This becomes the *Active Profile* definition for editing.
Copy	Lets you add a new wire profile name to the list. The profile is a copy of the existing active profile with the new name that you provide.
Delete	Lets you remove the active wire profile from the design. ⚠ You cannot delete the wire profile if it is currently assigned to one or more bond wires in the design.
Rename	Lets you rename the active profile. Renaming a profile will update all wires in the database with the profile. If you rename a vendor-certified wire bond profile, the profile will be marked as uncertified, as the vendor will be unable to locate the profile in their list of definitions.
Master Definitions	Specifies a list of all the predefined choices of profile libraries (vendor-supplied or developed by your internal CAD group) listed alphabetically and a *Browse* item that allows you to select a *Master Definitions* file that resides outside the TECHPATH. When you choose *Browse*, the tool displays a file browser to allow you to select the definitions file from disk, starting in the current working directory.

Save	Lets you save the current wire profile settings to a specified file. Typically, only the library group of your company will use this part of the Wire Profile Editor. If you click this button and your current master definitions are vendor-defined, you are prompted with the Save As dialog box to save as a new file name. This action causes all profiles to be flagged as uncertified. To restore certification, you must change back to the vendor's master profile definitions and allow the tool to refresh all profiles from that file.
Purge Unused Profiles	Purges all profiles that are not being used by wires in the current design.
Definition	
Direction	Specifies whether this profile defines a *Forward Bond* (wire runs from die pad to bond finger) or a *Reverse Bond* (wire runs from bond finger to die pad). The default setting is *Forward Bond*.
Material	Specifies the wire material.
Diameter	Specifies the wire diameter. Use a positive integer.
Refresh from Master	Click this button to return to the original specification (name is displayed in the Master Definitions field) if you have customized a wire profile in the current design or suspect that the master definitions have been updated since you last loaded the profiles from the library. If the profile is not defined in the Master Definitions field, this button is disabled.
Movement Type	This section describes the movement type with the horizontal and vertical planes used for each step.
Horizontal	Provides a menu with these options to specify the movement type in the horizontal plane (X/Y axes along the length of the wire): • Length – Specifies a constant distance movement along the horizontal wire from the previous point in the model. • Percent – Specifies the percentage of the difference between the wire start and end in the horizontal plane. Use values between 1 and 100. The tool supports decimals. • Angle – Specifies a change in angle of the given amount, with positive moving away from the substrate and negative moving towards it. • Switch – Indicates to the tool that it should take the next point from the other end of the wire. For example, if you are moving from the beginning of the wire, the next point starts at the other end of the wire.
Value	Lets you specify the value associated with the *Horizontal* movement type.

Vertical	Provides a menu with these options to specify the vertical movement type: • Length – Specifies a constant distance movement vertically from the previous point in the model. • Percent – Specifies the percentage of the elevation difference between the wire start and end. • Angle – Specifies a change in angle of the given amount, with positive moving away from the substrate and negative moving towards it. • Switch – Indicates to the tool that it should take the next point from the other end of the wire. For example, if you are moving from the beginning of the wire, the next point starts at the other end of the wire.
Value	Lets you specify the value associated with the vertical movement type.
	⚠ If you set the `wire_profile_ui_points_in_rows` user preference environment variable, the step appears in a row instead of a column.
Example	
Sample Start Height	*Specifies the starting height or the distance from the top of the substrate (used for the graphical example). By default, this setting is 200 UM to simulate an average die height. You can change the default value in the Wirebond section of the User Preferences dialog box.*
Sample Length	*Specifies the wire length to use for drawing the graphical example. This default setting is the maximum wire length constraint value in the current database. You can set a default value in the Wirebond section of the User Preferences dialog box.*
Profile Certified By Bitmap	If the active profile is certified by a vendor, the vendor's logo appears in this space. Note that any edits to the profile definition remove this certification.
Current example	This graphical panel illustrates the approximate look of the wire profile as currently specified. Because each bond wire that uses this profile may have a unique length, this visualization is an approximation only, based on the samples given.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Setting Wire Bond Parameters

Follow these steps to set wire bond parameters:

1. Early in the wire bond design process, run the `wirebond settings` command.
   The Wire Bond Settings dialog box appears.
2. Complete the parameters in the dialog box.
3. To edit constraints, use the Constraint Manager (cmgr).
   If you are in Feasibility mode (click the *Feasibility* button), click the *View/Edit constraints* button.
   The Global Wire Bond Constraints dialog box appears.
4. Complete the parameters for feasibility purposes only and click *OK*. When you are setting constraints, Cadence recommends that you use Constraint Manager.
5. To configure automatic bond finger label generation in the Wire Bond Settings dialog box, check the *Label bond fingers* box, and click the *Configure* button.
   The Bond Finger Labels Dialog Box appears.
6. Complete the parameters and click *OK*.
7. Click *OK* in the Wire Bond Settings dialog box.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Adding a Wire Profile

Follow these steps to add wire profiles:

1. Run the `wirebond settings` command.
2. From the Wire Bond Settings dialog box, click *View/Edit wire profiles*.
   The Wire Profile Editor appears. If this is the first time using the editor, default settings, based on your design, appear.
3. Click *Add and type in a new Profile name in the dialog box; then click OK.*
4. Specify the profile direction in the *Direction* field: *Forward Bond* or *Reverse Bond*.
5. Specify the material of the wire and its diameter.
6. Modify the default steps as desired, using the information described in the Wire Bond Profile Editor.
   The graphical representation changes as you change type of movement and values.
7. To insert a new step in the model, click in the *Step* number column for a specified point, right-click and choose *Insert* from the pop-up menu.
   Another step appears to the right.
8. Then click the drop-down menus for the *Horizontal* and *Vertical* components to specify the types of movement.
9. Be sure to enter the values for both horizontal and vertical components.
10. To remove a point, right-click in the *Step* number column for that point and choose *Remove* from the pop-up menu.
11. When finished, click *OK.*

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Initializing Your Wire Profiles

To initialize wire profiles from an `.xml` file:

a. Click the drop-down list in the *Master Definitions* field and choose *Browse* from the list.

b. Then click on the `.xml` file that you want to open. The directory path appears in the *Master Definitions* field and this message appears:

   Refresh existing database profiles from XML file? If you answer no, profiles defined in both the XML file and database will keep the current database definition.

c. Click *Yes*.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Deleting a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Copying a Wire Profile

Follow these steps to copy wire profiles:

1. Run the `wirebond settings` command at the console window prompt.
2. From the Wire Bond Settings dialog box, click *View/Edit wire profiles*.
   The Wire Profile Editor appears.
3. Select the profile that you want to copy from the *Active Profile* list.
   The settings for the specified profile appear in the dialog box.
4. Click *Copy*.
5. Enter the *Profile name* in the dialog box and click *OK*.
   The settings are part of the new profile.
6. Modify settings of the newly copied profile as necessary (See Editing a Wire Profile).
7. When finished, click *OK*.
   Clicking *OK* saves the new, copied profile to the database. Click *Save* to save the profile to a file on disk.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Editing a Wire Profile
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Deleting a Wire Profile

Follow these steps to delete wire profiles:

1. Run the `wirebond settings` command at the console window prompt.
2. From the Wire Bond Settings dialog box, click *View/Edit wire profiles*.
   The Wire Profile Editor appears.
3. Select the profile that you want to delete from the *Active Profile* list.
   The settings for the specified profile appear in the dialog box.
4. Click *Delete*.
   The specified profile name is removed from the *Active Profile* list. You cannot delete the wire profile if it is currently assigned to one or more bond wires in the design.
5. When finished, click *OK*.
   Clicking *OK* deletes the new, copied profile from the database. Click *Delete* to delete the profile from a file on disk.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Refreshing a Wire Profile's Definition from the Master Definitions File
- Purging Unused Wire Profiles from a Design

Editing a Wire Profile

Follow these steps to edit wire profiles:

1. Run the `wirebond settings` command at the console window prompt.
2. From the Wire Bond Settings dialog box, click *View/Edit wire profiles*. The Wire Profile Editor appears.
3. Select the profile that you want to edit from the *Active Profile* list
4. Follow steps 5 through 12 in the Adding a Wire Profile.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Purging Unused Wire Profiles from a Design

Refreshing a Wire Profile's Definition from the Master Definitions File

Follow these steps to refresh a wire profile definition:

1. From the Wire Profile Editor dialog box, click *Refresh from Master*.
   Settings from the original specification appear in the dialog box.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Predefined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile

Purging Unused Wire Profiles from a Design

Follow these steps to purge unused wire profiles from your design:

1. From the Wire Profile Editor dialog box, click *Purge Unused Profiles*.
   A confirmer dialog box appears.
2. Click *Yes*.
   The unused profiles are removed from the design.

Related Topics

- wirebond settings
- Wire Bond Settings Dialog Box
- Bond Finger Labels Dialog Box
- Global Wire Bond Constraints Dialog Box
- Pre-defined Wirebond Settings Dialog Box
- Wire Bond Profile Editor
- Setting Wire Bond Parameters
- Adding a Wire Profile
- Initializing Your Wire Profiles
- Copying a Wire Profile
- Deleting a Wire Profile
- Editing a Wire Profile

wirebond soldermask create

The `wirebond soldermask create` command lets you quickly and easily create soldermask shapes that encompass a set of bondpads. The command creates a single shape that clears all the bondpads to the extent of the clearance value that you entered. It also rounds the two ends of the shape.

⚠ The tool now remembers the settings used by storing them with the design.

Related Topics

- Creating Soldermask Shapes for Bondpads Automatically

Wirebond Soldermask Create Command: Options Panel

Access Using

- *Menu Path: Manufacture – Create Bond Finger Soldermask*

Delete existing solder mask	When enabled, deletes any existing solder mask on the same layer and at the same location as the new solder mask being created.
Use top/bottom solder mask layer	Lets you toggle between creating the solder mask on SOLDERMASK_TOP or SOLDERMASK_BOTTOM. Usually this depends on whether your wirebonded IC is mounted in the chip-up or chip-down orientation.
Mask type	Select either *Finger*, the default, or *Ring*.
Distance from finger edge	Lets you specify the clearance to the finger pad for the outside, inside or ends of the fingers. You can specify a negative value to have only a portion of the pad exposed through the soldermask layer. Available for Finger.
Minimum Allowed Indent Width	Allows indents of the specified width along the edge of a soldermask to remain unfilled after widths of a smaller value are filled in. Available for Finger.
Minimum Allowed Tier Spacing	Combines finger tiers if they are closer than the specified value. For example, if there are two rows of bond fingers and there is a gap between the generated soldermask, setting this value ensures the gap is filled for gaps less than the specified value. Available for Finger.
Mask Opening Dimensions	Lets you specify the width and height of the mask opening. Available for Ring.
Rotate mask to match object rotation	Check to rotate the mask to match the rotation of the object. You can also select the *Snap to nearest 90 degree rotation* option. Available for Ring.

Trim mask opening to ring edge	Check to trim the mask to within the specified clearance. Available for Ring.

Creating Soldermask Shapes for Bondpads Automatically

To automatically create soldermask shapes for bondpads, follow these steps:

1. Run the `wirebond soldermask create` command.
2. Set the *Options* panel controls as described above.
3. Choose the bondpads to create a soldermask for.
4. Choose *Done* from the right-button pop-up to complete the command.

Related Topics

- wirebond soldermask create

wirebond tack point

The `wirebond tack point` command lets you move the start point of the wire within the die pad or the endpoint of a bond wire within the bond finger area.

For additional information on flows and wire bond tools, see the *Routing User Guide* in your documentation set. For information on design tasks, see Generic Wire Bond Design Tasks under the `wirebond select` command.

Related Topics

- Running the Wirebond Tack Point Command
- Automatically Spreading All Wires on Fingers
- Moving the Tack Point Along the Pad Axis
- Moving the Tack Point Freely Within the Pad
- Copying an Existing Tack Point Offset
- Removing a Tack Point Offset

Wirebond Tack Point Command: Options Panel

Access Using

- *Menu Path: Route – Wire Bond – Tack Point Move*

The following table describes the parameters that you complete in the *Options* panel of the Control Panel when you run the `wirebond tack point` command.

Wire end to move	Determines the point of the selected bond wires that you are adjusting. The default setting is to adjust the wire end on the bond finger.
Start (pin)	Click this button to adjust the start point of the selected wires. For die-to-die bond wires, the tool adjusts the end on the pin higher up in the die stack.
End (finger)	Click this button to adjust the end point of the selected bond wires. In cases where you have an interposer bond finger-to-substrate bond finger, the tool edits the substrate bond finger's wire end point. This is the default setting.
Axis of movement	When picking the die pad ends of a bond wire, the *axis* for movement may differ depending on your application.
Along axis /die edge	If you click this button, incremental and interactive movement is parallel to the die edge over which the bond wire crosses. This is useful when bonding multiple wires to the same pin pad. This is the default setting.
Orthogonal to axis/ die edge	If you click this button, incremental and interactive movement is perpendicular to the die edge over which the bond wire cross. This is useful when moving a single wire in or out on a larger die pad.
Auto spread all wires on finger	
Movement along pad axis	Click this button to activate either of the two axis-based movements: *Interactive* or *Incremental*. Choosing *Incremental* lets you specify the value by which the tool moves, in the text box below.
Free movement inside pad	Click this button to freely move within the pad.

Copy existing tack point offset	Click this button to select bond wires and assign their tack points to the same offset (from the bond finger origin) as another existing tack point. After you select this option, select a source from which you want to copy the offset, then select one or more target bond wires to which you want to copy the offset. The tool remembers the source wire for copying until you right-click and choose *Next* from the popup menu. You can copy to multiple wires without using the *Temp Group* option or re-selecting the source wire every time.
Remove tack point offset(s)	Click this button to move all selected bond wire tack points back to the origin of the bond fingers and remove the tack point shapes.
Min distance from pad edge	Check the box and enter a value to specify a minimum distance from the pad edge when you are using any of these options: • Movement along pad axis • Free movement inside pad • Copy existing tack point offset

Related Topics

- Automatically Spreading All Wires on Fingers
- Moving the Tack Point Along the Pad Axis
- Moving the Tack Point Freely Within the Pad
- Copying an Existing Tack Point Offset
- Removing a Tack Point Offset

Running the Wirebond Tack Point Command

⚠ When using the `wirebond tack point move` command when there are multiple wires at the same location and you click a wire that you do not want, you can right-click and choose *Reject* from the po-up menu and then click an alternate wire.

Related Topics

- wirebond tack point
- Moving the Tack Point Along the Pad Axis
- Moving the Tack Point Freely Within the Pad
- Copying an Existing Tack Point Offset
- Removing a Tack Point Offset

Automatically Spreading All Wires on Fingers

To spread all wires on fingers automatically:

1. Run the `wirebond tack point` command.
   You are prompted with this message:

   Select bondwire(s).
   Bond Wires is set in the Find Filter.
2. From the *Options* panel in the Control Panel, choose one of the options for *Wire end to move: Start (pin)* or *End (finger).*
3. Choose an option under *Axis of movement* either along the axis/die edge or orthogonal to the axis/die edge.
4. Click *Auto spread all wires on finger*.
   Fingers is now set in the Find Filter.
5. Choose the bond finger on which you are spreading the wires.
   The wires automatically spread.
6. Right-click in the Design Window and choose *Done*.

Related Topics

- wirebond tack point
- Wirebond Tack Point Command: Options Panel
- Moving the Tack Point Freely Within the Pad
- Copying an Existing Tack Point Offset
- Removing a Tack Point Offset

Moving the Tack Point Along the Pad Axis

To move a tack point along the pad axis:

1. Run the `wirebond tack point` command.
   You are prompted with this message:

   Select bondwire(s).
   Bond Wires is set in the Find Filter.
2. From the *Options* panel in the Control Panel, choose one of the options for *Wire end to move: Start (pin)* or *End (finger).*
3. Choose an option under *Axis of movement* either along the axis/die edge or orthogonal to the axis/die edge.
4. Click *Movement along pad axis.* Then either click *Interactive* or *Incremental*. If you choose *Incremental*, enter a value in the text box.
5. Click on one or more bond wires.
   If you chose *Incremental*, the bond wires automatically move according to the specified value.
6. If you chose *Interactive*, determine the point where you are adjusting the bond wire.
   The ending or starting points of the bond wires dynamically track the mouse movement.
7. Click the mouse to complete the operation.
8. Perform steps 3 through 10 until you have made all your changes.
9. Right-click in the Design Window and choose *Done*.

Related Topics

- wirebond tack point
- Wirebond Tack Point Command: Options Panel
- Running the Wirebond Tack Point Command
- Copying an Existing Tack Point Offset
- Removing a Tack Point Offset

Moving the Tack Point Freely Within the Pad

To move a tack point freely within a pad:

1. Run the `wirebond tack point` command.
   You are prompted with this message:

   Select bondwire(s).
   Bond Wires is set in the Find Filter.
2. From the *Options* panel in the Control Panel, choose one of the options for *Wire end to move: Start (pin)* or *End (finger).*
3. Click *Free movement inside pad*.
4. Specify the minimum distance from the pad edge by checking the *Min distance from pad edge* box and entering a value in the text box.
5. Click on a bond wire.
   You are prompted with this message:

   Enter point:
6. Determine the point where you are adjusting the bond wire.
   The ending or starting points of the bond wire dynamically tracks the mouse movement.
7. Click the mouse to complete the operation.
8. Perform steps 3 through 7 until you have made all your changes.
9. Right-click in the Design Window and choose *Done*.

Related Topics

- wirebond tack point
- Wirebond Tack Point Command: Options Panel
- Running the Wirebond Tack Point Command
- Automatically Spreading All Wires on Fingers
- Removing a Tack Point Offset

Copying an Existing Tack Point Offset

To copy an existing tack point offset:

1. Run the `wirebond tack point` command.
   Bond Wires is set in the Find Filter.
2. From the *Options* panel in the Control Panel, choose one of the options for *Wire end to move: Start (pin)* or *End (finger).*
3. Click *Copy existing tack point offset.*
   You are prompted with this message:

   Select a bondwire with Tack Point.
4. Specify the minimum distance from the pad edge by checking the *Min distance from pad edge* box and entering a value in the text box.
5. Click on the bond wire with tack point that you want to copy.
   You are prompted with this message:
   Select bondwire(s) to copy Tack Point.
6. Select the bond wires to which you are copying.
   The layout tool adjusts the bond wires.
7. Right-click and choose *Next* in the pop-up menu to pick a new master bond wire from which to copy the offset.
8. Perform steps 2 through 8 until you have made all your changes.
9. Right-click in the Design Window and choose *Done*.

Related Topics

- wirebond tack point
- Wirebond Tack Point Command: Options Panel
- Running the Wirebond Tack Point Command
- Automatically Spreading All Wires on Fingers
- Moving the Tack Point Along the Pad Axis

Removing a Tack Point Offset

To remove a tack point offset:

1. Run the `wirebond tack point` command.
   Bond Wires is set in the Find Filter.
2. From the *Options* panel in the Control Panel, choose one of the options for *Wire end to move: Start (pin)* or *End (finger).*
3. Click *Remove Tack Point (offsets)*.
   You are prompted with this message:

   Select bondwire(s) with tack point(s).
4. Click the bond wires with the tack points that you want to remove.
   The layout tool moves all the tack points back to the origin of the bond fingers or die pins.
5. Right-click in the Design Window and choose *Done*.

Related Topics

- wirebond tack point
- Wirebond Tack Point Command: Options Panel
- Running the Wirebond Tack Point Command
- Automatically Spreading All Wires on Fingers
- Moving the Tack Point Along the Pad Axis
- Moving the Tack Point Freely Within the Pad

wirebond unlock

The `wirebond unlock` command unlocks all bond wires and bond pads or fingers from the die components to which they are attached in your design.

By default, bond wires and bond fingers and the connected die symbol are in the locked state; Bond wires and bond fingers are created linked to the die symbol with a parent and child relationship between the bond pads or fingers and bond wires with the symbol they are connected. The parent and child relationship or locking allows editing commands such as *Move*, *Delete*, *Rotate* to treat the wires and fingers as part of the die symbol, acting on them all as a single unit. For example, if you move a die symbol while locked, the bond wires and bond fingers connected to the die symbol are also moved. Similarly, deleting or unplacing the die symbol in the locked state rips up the bond items.

Unlocking wire bonds causes the following conditions:

- Bond wires rubberband when you move a die.
- Bond wires do not get ripped up when you delete or unplace a die.

> ⓘ You cannot undo the `wirebond unlock` command once you have saved the design.

Running the Wirebond Unlock Command

Follow these steps to run the `wirebond unlock` command:

1. In the command window, enter `wirebond unlock`.
   A conformation window appears explaining the feature and asking if you want to continue.
2. Click *Yes* to unlock the die symbols and their connected wires and fingers in the open design.

wirebond uprev

The `wirebond uprev` command lets you uprev a design that has bond wires using the old model or a design that did not uprev during the conversion from Release 15.7 to Release 16.0. For example, you need to run this command if you have a design with bond wires that do not have the BOND_WIRE property or bond fingers that do not have the BOND_PAD property attached. When you open such a design in Release 16.0, the tool does not automatically convert the design. Based on the log file, you need to add the property to the bond wires and bond fingers and then run this command.

Access Using

- *Menu Path: Route – Wire Bond – Uprev*

Running the Wirebond Uprev Command

Perform this step to run the `wirebond uprev` command:

1. Choose *Route – Wire Bond – Uprev* (`wirebond uprev`).
   The design is converted.

wire bond via estimation

The `wire bond via estimation` command lets you determine if there is enough room near a single die or die stack for vias before routing a wire bond package. Without an estimator, you might start routing, and have to move bond fingers when you find out that there is not enough room for the vias.

For additional information about the wire bond via estimator, see the *Routing User Guide* in your documentation set.

Related Topics

- Set Via-Pad Size Dialog Box
- Running the Wirebond Via Estimation Command

Wire Bond Via Estimation Dialog Box

Access Using

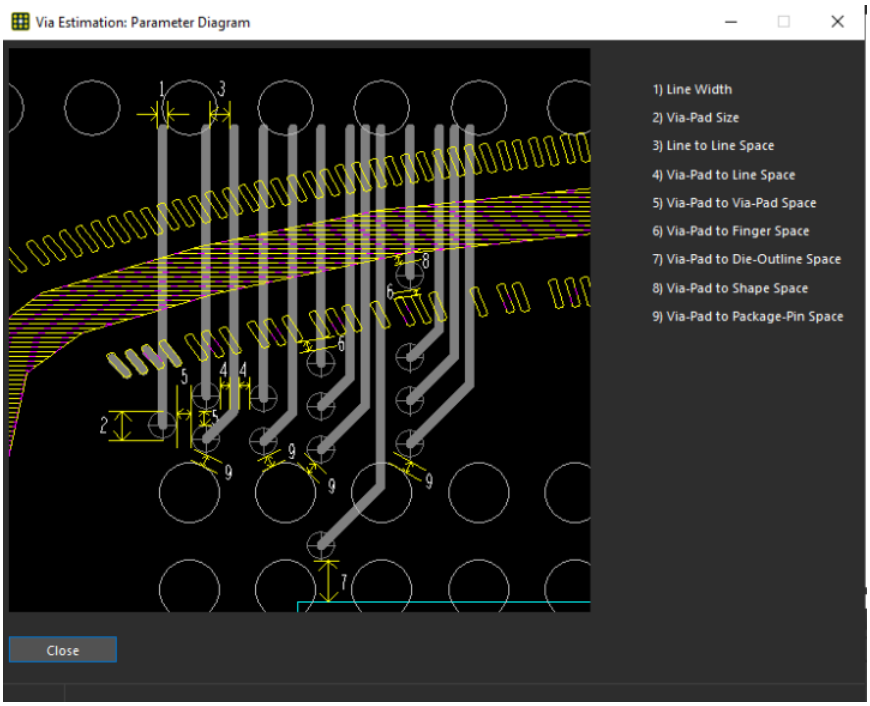
- *Menu Path: Route – Wire Bond Via Estimation*

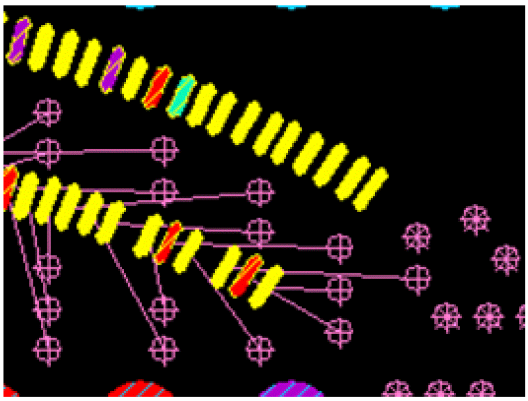
The Wire Bond Via Estimation dialog box appears when you run the `wire bond via estimation` command. Some of the parameters described below have a number listed. This number points to an example of the parameter displayed on clicking *Parameter Diagram*.

The first time that you use this dialog box, the line width and spacing options default to values from the design constraint set, as appropriate. You can modify the values as needed.

	Die Ref Des (for APD only)	Specifies the name of the wire bond die on which you want to run the `wire bond via estimation` command. The tool lists all the wire bond die found in the package in the pull-down menu.
	Die Stack (for SiP only)	Specifies the wire bond die-stack for the via estimation. The tool lists all the wire bond die-stacks found in the package in the pull-down menu.
1	*Line Width*	Specifies the line width that the tool uses for simulated lines that exit vias on the escape layer. This value is initially set to the *Min Line Width* value that you set for the top *Etch* subclass of the design constraint set.
2	*Via-Pad Size*	Specifies the via pad size. This value is initially set to the via pad size of a thru-hole padstack found on the top *Etch* subclass in the design. If there is more than one thru-hole padstack, the tool chooses one arbitrarily. If none are found, the value is 0.
	Padstack	Click this button to display the Set Via-Pad Size dialog box.
3	*Line to Line Space*	Specifies the line-to-line spacing for the simulated lines on the escape layer. This value is initially set to the *Line to Line* spacing value that you set for the top *Etch* subclass in the design constraint set.
4	*Via-Pad to Line Space*	Specifies the space between the via pad and the simulated lines on the escape layer. If the via list of the design constraint set contains a thru-hole via, then the default is the value of *Thru Via To Line* that you set for the top *Etch* subclass in the design constraint set. *Otherwise, the default is the value of* the *BB Via To Line* constraint.

5	*Via-Pad to Via-Pad Space*	Specifies the spacing between via pads. If the via list of the design constraint set contains a thru-hole via, then the default is the value of *Thru Via To Thru Via* that you set for the top *Etch* subclass in the design constraint set. *Otherwise, the default is the value of* the *BB Via To BB Via* constraint.
6	*Via-Pad to Finger Space*	Specifies the minimum distance from the via pad markers to all bond fingers. If the via list of the design constraint set contains a thru-hole via, then the default is the value of the *Thru Via To Bond Finger constraint* that you set for the top *Etch* subclass in the design constraint set. *Otherwise, the default is the value of* the *BB Via To Bond Finger* constraint.
7	*Via-Pad to Die-Outline Space*	Specifies the shortest distance that the tool can place a via marker from the die outline. This value is initially set to 0.
8	*Via-Pad to Shape Space*	Specifies the minimum distance from the via pad marker to a shape, such as power and ground rings. If the via list of the design constraint set contains a thru-hole via, then the default is the value of the *Thru Via To Shape constraint* that you set for the top *Etch* subclass in the design constraint set. *Otherwise, the default is the value of* the *BB Via To Shape* constraint.
9	*Via-Pad to Package-Pin Space*	Specifies the minimum space from the via pad markers to the pins. If the via list of the design constraint set contains a thru-hole via, then the default is the value of the *Thru Via To SMD Pin constraint* that you set for the top *Etch* subclass in the design constraint set. *Otherwise, the default is the value of* the *BB Via To SMD Pin* constraint.
	Number of Escape Layers	Specifies the number of escape layers to use. This value is initially set to 1, but you can increase it if you want more escape layers. ⚠ Even if this value is greater than 1, the tool draws the escape lines only on the Via_Estimation_Top subclass.
	Include Voltage Nets	If you uncheck this box, then the tool reduces the bond finger count in the report by the number of voltage bond fingers. No lines are drawn from voltage bond fingers to via markers. The number of via markers is unaffected by this option. A bond finger is a *voltage* type if its net has the VOLTAGE property, or if its die pin is either POWER or GROUND. By default, the button is checked.

	Parameter Diagram	Click this button to display an example highlighting some of the parameters. The following figure shows examples of the size and spacing parameters listed in the Wire Bond Via Estimation Dialog Box.  Numbers shown in the figure correspond to the numbered option in the dialog box. For example, 1 is *Line Width,* 2 is *Via-Pad Size,* and so on. Escape clines are shown in this example to clarify the meaning of the parameters. Lines from the bond fingers to vias are not shown. On the north side of the die stack, in this example, columns are created left to right, with vias in each column created bottom to top. The estimator creates additional columns of vias (not shown) until the right edge of the die or die stack is reached.
	Display Options	
	Show Escape Lines	Check this box to display the simulated clines on the escape layer in the design once the tool completes the via estimation. The lines shown adhere to the specified line-width and spacing values. This option helps you verify that the via estimation adheres to the specified parameters. You may want to turn off this option once you are confident that the algorithm is computing the escape wires accurately. The default value is enabled.

	Show Finger to Via Lines	Click this button to show the lines from the vias markers to the bond fingers once the tool completes the via estimation. These lines are on the Via_Estimation_Top subclass. The default value is enabled. The tools sorts the lines between bond fingers and via markers so that if possible, the lines do not cross. It gives the via markers that are nearest to the bond fingers priority, so that if extra via markers exist, they tend to be those in the corners farthest from the bond fingers. If there are more via markers than bond fingers, then the extra via markers do not show a connection to a bond finger, but show an "X" overlapping the via marker. This sample picture shows lines between bond fingers and via markers. The extra via markers (toward the right) are shown as circles with "+" and an overlapping "X". 
	Run Via Estimation	Click this button to delete any previous markers, and create new via markers. A report displays the statistics of bond fingers versus via markers.
	Delete Via Markers	Click this button to delete any previous via markers. Do this when you have completed routing and no longer need the via markers.
	OK	Click this button to exit the dialog box. The tool preserves the option values and the Via_Estimation_Top subclass.
	Cancel	Click this button to exit the dialog box. The tool reverts the option values and the Via_Estimation_Top subclass back to the state before you ran the command.
	Help	Provides help about the `wire bond via estimation` command.

Related Topics

- Running the Wirebond Via Estimation Command

Set Via-Pad Size Dialog Box

When you click *Padstack* in the Wire Bond Via Estimation dialog box, the Set Via-Pad Size dialog box appears.

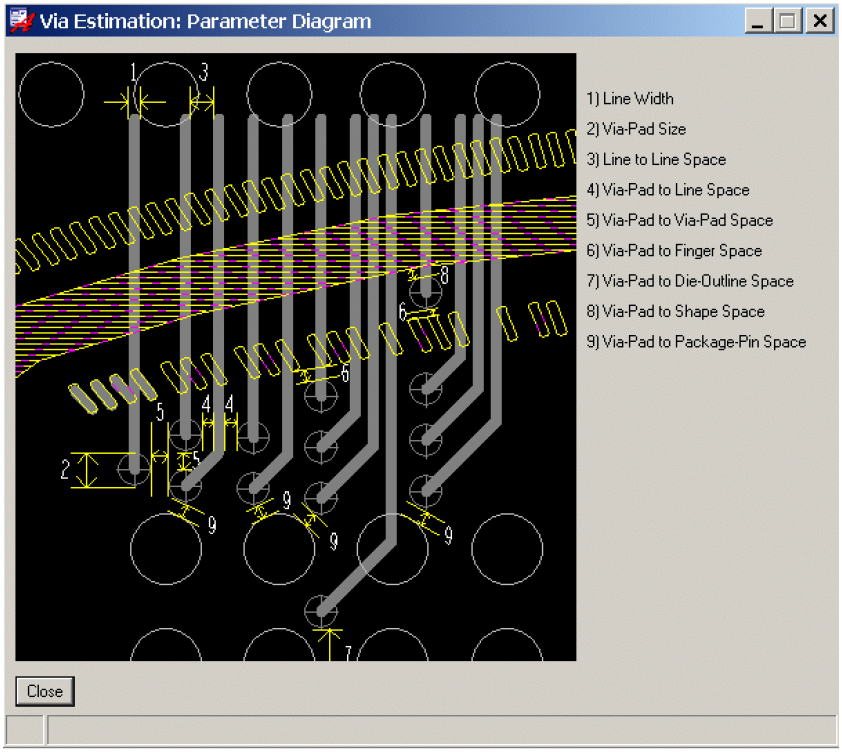
Padstack	Specifies the database padstack that you want to use for the via marker pads. Once you select a padstack from the pull-down list, the *Via Pad Size* field is automatically set to the chosen padstack's pad size set, as defined in the Wire Bond Via Estimation dialog box for the top *Etch* subclass.
OK	Saves the changes and exits the dialog box.
Cancel	Closes the dialog box without making any changes.
Help	Provides help on this command.

Parameters Example

The following figure shows examples of the size and spacing parameters listed in the Wire Bond Via Estimation Dialog Box.

Numbers shown in the figure correspond to the numbered option in the dialog box. For example, 1 is *Line Width,* 2 is *Via-Pad Size,* and so on.

Escape clines are shown in this example to clarify the meaning of the parameters. Lines from the bond fingers to vias are not shown. On the north side of the die stack, in this example, columns are created left to right, with vias in each column created bottom to top. The estimator creates additional columns of vias (not shown) until the right edge of the die or die stack is reached.



Related Topics

- wire bond via estimation

Running the Wirebond Via Estimation Command

After you create your bond fingers but before you perform routing:

1. Choose *Route – Wire Bond Via Estimation from the menu*.
   The Wire Bond Via Estimation dialog box appears with default values listed from the design constraint set.
2. Modify the parameters.
3. Click *Run Via Estimation*.
   The tool creates via markers indicating where you can place vias. A report displays the results.
4. Modify the parameters and run the estimator again.
   -or-
   Exit the dialog box and begin routing, or move the bond fingers.
   Once the estimator indicates success, you can begin routing.

Related Topics

- wire bond via estimation
- Wire Bond Via Estimation Dialog Box

workflow

The `workflow` command displays the Workflow Manager, which a master designer uses to manage all sections or partitions of the primary (master) design, after they are created with *Place – Design Partition – Create Partitions* (`partition` command). For each partition created, an entry appears in the Workflow Manager. An extension of `.dpf` is appended to PCB Editor partition files. An extension of `.dpm` is added to APD partition files and `.dps` to SiP partition files.

A master designer exports the partitions to partition designers, who are restricted to independent copies of the database with a limited command set. Multiple designers can then work concurrently on their respective partitions to complete the design.

However, the master designer can designate certain nets as soft, which means that the partition designer can pick and route these nets even if they extend beyond the boundary of the active assigned partition. Soft nets are highlighted in the owner's partition database but are dimmed and read-only in all other partitions.

The master designer can allow partition designers to edit Physical, Spacing, and Electrical Constraints in their respective partitions.

⚠ Design level constraint editing is not permitted in partition databases.

You can create partitions by layer to restrict partition designers to specific layers instead of all the layers in the Z axis. This allows partition designers to work directly over and under each other.

The master designer imports or refreshes modified partitions into the master design. No automatic updates to the master design occur, nor do partition databases update in real time.

During design partitioning, a master designer may attach the LOCKED property to symbols and modules to prevent partition designers from making modifications. For example, symbols and modules with the LOCKED property prevent partition designers from modifying individual symbol or module parts (such as pins, padstacks, and so on) when placed, although they can delete, move, or rotate the symbol or module as a single object and edit any attached text.

Partition designers choose when to refresh information from the master database or other partitions for a current view of the design. Updates occur from saved databases only. The master designer can choose to refresh the master design, which allows the partition designers to continue working on their current view.

Importing a partition into the master database stops the designer assigned to that partition from working and frees it in the master database so the master designer may modify it. For the partition designer to resume work, the master designer must export the partition again. The old partition database then becomes obsolete. For more information on design partitioning, see *Partitioning a Design* in the *Placing the Elements* user guide in your documentation set.

Related Topics

- Importing a Partition into a Master Database
- Exporting a Partition from a Master Database
- Refreshing a Partition
- Retracting a Partition
- Previewing Partitions Prior to Export from The Master Database
- Appending Notes to a Master or Partition Database
- Reporting Connection Statistics
- Recovering an Obsolete Partition
- partition

Workflow Manager

Access Using:

- *Menu Path: Place – Design Partition – Workflow Manager*

Select	Choose one or more partitions to import, export, refresh, delete, retract, or preview if you are a master designer; if you are a partition designer, refreshing is the only action you can perform.
Name	Lists the existing partition names. On the initial creation of a partition, its name defaults to PARTITION_2, PARTITION_3, and so on. The MASTER_DESIGN is always PARTITION_1. Once the partition exists, but prior to exporting it, you can change its name. Two pre-defined partitions—SILKSCREEN_TOP_BOTTOM DIMENSION_DRAFTING—are created automatically and cannot be changed or deleted.
Location	For the master design, the system automatically generates the path, which defaults to the current working directory. For a partition, the system creates a directory and a partition file beneath it named PARTITION_2, PARTITION_3, and so on. An extension of `.dpf, .dpm,` or `.dps` is appended to the partition files. This field is read-only until you export partitions. Once exported, change the partition path by left-clicking to display a browser from which a master partition may search for an exported partition whose default path may have changed; a partition may search for the master partition or another partition in the design. A red background in this cell indicates the current path is invalid.
User	Defaults to the login that the master designer assigns to the partition. The login must be valid to support the design tool's e-mail program. Each time you invoke Workflow Manager, the design tool partially opens each partition to populate this field.
Start Layer	For partitioning by layer, specifies the start layer of a horizontal partition (as opposed to all layers in the Z axis). You can assign one layer or multiple consecutive layers. For example, you can specify layer 2 as both the *Start* layer and *Stop* layer. Or you can assign multiple consecutive layers and assign layer 2 as the *Start* layer and layer 4 as the *Stop* layer. This means that the partition designer is responsible for layers 2, 3, and 4. The partition command is disabled once you export a partition. You can also assign layers using the `partition` command.

Stop Layer	For partitioning by layer, specifies the end layer of a horizontal partition (as opposed to all layers in the Z axis), You can assign one layer or multiple consecutive layers. For example, you can specify layer 2 as both the *Start* layer and *Stop* layer. Or you can assign multiple consecutive layers and assign layer 2 as the *Start* layer and layer 4 as the *Stop* layer. This means that the partition designer is responsible for layers 2, 3, and 4. The partition command is disabled once you export a partition. You can also assign layers using the `partition` command.
Overlap	Indicates whether partitions overlap; if so, YES displays in red. Overlapped partitions cannot be exported. The master designer must eliminate overlaps prior to exporting partitions.
Status	Specifies the state of the database as *Master, Inactive, Active,* or *Exported*. An *Inactive* status indicates an unexported database, or one exported and then re-imported without modification.
Progress	Specifies the state of a partition as *New, In Progress*, or *Complete*. Each time you invoke Workflow Manager, the design tool partially opens each partition to populate this field.
Unplaced	This field is read-only and indicates the number of symbols that become unplaced in an exported partition when a partition designer moves them outside the partition.
% Routed	This field is read-only and indicates the percentage of connections routed in an exported partition.
Append Note	Enter additional instructions, guidelines, comments, and design information for one or more unexported partitions. Subsequent notes are appended to previous comments and remain as a permanent record during a partition's existence. Each time you invoke Workflow Manager, the design tool partially opens each partition to populate the *Notes* field.
Notes	This field is read-only and displays a chronology of instructions, guidelines, comments, and design information. Each time you invoke Workflow Manager, the design tool partially opens each partition to populate this field.
Import	Integrates the chosen partition databases into the master database when a partition designer finishes or tasks remain in progress. You must save a partition database to disk and make it accessible either in the directory to which it was originally exported, or through a path the partition designer specifies. Importing a partition renders it invalid, and you cannot re-import it into the master design again.
Export	Duplicates a portion of the master database specified by the master designer and creates a partition from it for the assigned partition designer. Before a partition designer works on an assigned partition, the master designer must export it. The master designer may export as many partitions as required. Once the master designer exports a partition, the partition designer automatically receives e-mail notification, unless you enable *Suppress Mail*.

Mail	Click to display the design tool's internal e-mail interface (`mail` command) to facilitate communication among the designers.
Refresh	Updates the master database with the partition, allowing the master designer to determine whether the partition modifications function correctly in the master design and with those in other partitions. The partition designer can continue working. Partition designers can refresh information from the master database or from other partitions for the most current view of the design.
Select All	Choose all partitions to import, export, refresh, delete, retract, or preview if you are a master designer. If you are a partition designer, refreshing is the only action that you can perform.
Defer DRC on Import	Check this box to defer DRC update when importing partitions into the master database. Use this option to increase performance or speed and when DRC update is not important. Uncheck this box during final verification.
Report	Generates a Design Partition Report for the specified partitions, including the names and number of partitions, their status, locations, assigned designers, notes, and connection statistics.
Preview	Generates a Partition Preview Report for a chosen partition prior to export that specifies its contents, including refdes, components, and package names, which highlight on the board. The *Is Shared* column indicates whether a component straddles the boundary of one or more partitions, possibly including the master design.
ViewLog	Click for detailed information in the `workflow_manager.log` concerning the export, import, or retract processes; deleted partitions; database error messages resulting from the export, import, retract, delete, or refresh processes; and e-mail suppression or failures. You can also obtain pertinent information from the Status bar on the Workflow Manager.
Edit Cns	Select to make constraints editable. If you select this option, partition designers will be able to edit Physical, Spacing, and Electrical constraints in the partitions assigned to them. Note that design level constraints cannot be edited by partition designers.
Suppress Mail	Prevents the design tool from automatically sending e-mails to designers when you export or retract partitions.
Recovery Mode	Choose to allow an obsolete partition file to be imported into the master database. During team design, this may occur when a partition file becomes obsolete when the master designer re-imports it, but the partition designer continues work.
Soft Boundary	Select to enable partition designers to move components or route signals outside their boundaries. When soft boundaries are enabled those components moved outside will be saved during the export.

Delete	Removes chosen partition boundaries. You cannot use *Edit – Delete* to remove partition boundaries. To modify partition boundaries, use *Shape – Edit Boundary* (`shape edit boundary` command).
Retract	Click to cancel all exported partitions when partition contents change, rendering the exported partitions obsolete. The master designer notifies partition designers to stop working and delete their partition databases because retracted partitions cannot be imported into the master design. Retracting a partition renders it invalid, and you cannot import it into the master design again.
Apply	Click to accept modifications to partition-related parameters. Other modifications occur in memory only and are not committed to disk unless you choose *File – Save*.
OK	Terminates the command. Other modifications occur in memory only and are not committed to disk unless you choose *File – Save*.
Help	Displays help for this command.

Related Topics

- Exporting a Partition from a Master Database
- Refreshing a Partition
- Retracting a Partition
- Previewing Partitions Prior to Export from The Master Database
- Appending Notes to a Master or Partition Database
- Reporting Connection Statistics
- Recovering an Obsolete Partition
- mail

Importing a Partition into a Master Database

To import a partition to your master database, follow these steps:

1. Choose *Place – Design Partition – Workflow Manager* (`workflow` command).
2. Choose as many exported partitions as required by enabling the *Select* field.
3. Ensure that you update the partition with the latest modifications using *Refresh.*
4. Click *Import.*
5. Click *ViewLog* for information for detailed information in the `workflow_manager.log`. These details include the success or failure of the export, import, or retract processes; deleted partitions; database error messages resulting from the export, import, retract, delete, or refresh processes; and e-mail suppression or failures.

Related Topics

- workflow
- Refreshing a Partition
- Retracting a Partition
- Previewing Partitions Prior to Export from The Master Database
- Appending Notes to a Master or Partition Database
- Reporting Connection Statistics
- Recovering an Obsolete Partition

Exporting a Partition from a Master Database

To export a partition from your master database, follow these steps:

1. Choose *Place – Design Partition – Workflow Manager (*`workflow` command*).*
2. Choose as many partitions as required by enabling the *Select* field.
3. Modify the user, location, or partition name for the chosen partitions as necessary.
4. Create notes in the *Append Note* field as required.
5. Click *Apply* to save changes. (Information entered in the *Append Note* field appears in the *Notes* field when you click *Apply*.)
6. Click *Export.* The *Status* field for the chosen partitions changes to *Exported*.

Related Topics

- workflow
- Workflow Manager
- Retracting a Partition
- Previewing Partitions Prior to Export from The Master Database
- Appending Notes to a Master or Partition Database
- Reporting Connection Statistics
- Recovering an Obsolete Partition

Refreshing a Partition

Follow these steps to refresh a partition:

1. Choose *Place – Design Partition – Workflow Manager* (`workflow` command).
2. Choose as many partitions as required by enabling the *Select* field.
3. Click *Refresh.*
4. Click *ViewLog* for information concerning the process.

Related Topics

- workflow
- Workflow Manager
- Importing a Partition into a Master Database
- Previewing Partitions Prior to Export from The Master Database
- Appending Notes to a Master or Partition Database
- Reporting Connection Statistics
- Recovering an Obsolete Partition

Retracting a Partition

Follow these steps to retract a partition:

1. Choose *Place – Design Partition – Workflow Manager* (`workflow` command).
2. Choose as many partitions as required by enabling the *Select* field.
3. Verify the partitions' *Status* field is *Exported*.
4. Click *Retract* to cancel all exported partitions and return them to their original state. The partition's *Status* field displays *Retracted*.

Related Topics

- workflow
- Workflow Manager
- Importing a Partition into a Master Database
- Exporting a Partition from a Master Database
- Appending Notes to a Master or Partition Database
- Reporting Connection Statistics
- Recovering an Obsolete Partition

Previewing Partitions Prior to Export from The Master Database

Use the Partition Preview Report to determine that the partition contents are correct before you export it.

1. Verify that the partition's *Status* field is not *Exported*.
2. Enable the *Select* field for the partition.
3. Click *Preview.*
   The Partition Preview Report appears.

Related Topics

- workflow
- Workflow Manager
- Importing a Partition into a Master Database
- Exporting a Partition from a Master Database
- Refreshing a Partition
- Reporting Connection Statistics
- Recovering an Obsolete Partition

Appending Notes to a Master or Partition Database

To append notes to a database, follow these steps:

1. Choose *Place – Design Partition – Workflow Manager* (`workflow` command).
2. Enable the *Select* field for the partition.
3. Enter the information in the *Append Note* section.
4. Click *Apply*.

Related Topics

- workflow
- Workflow Manager
- Importing a Partition into a Master Database
- Exporting a Partition from a Master Database
- Refreshing a Partition
- Retracting a Partition
- Recovering an Obsolete Partition

Reporting Connection Statistics

To generate the design partition reopprt, follow these steps:

1. Choose *Place – Design Partition – Workflow Manager* (`workflow` command).
2. Enable the *Select* field for the partition.
3. Click *Report.*
   The Design Partition Report appears on the screen.

Related Topics

- workflow
- Workflow Manager
- Importing a Partition into a Master Database
- Exporting a Partition from a Master Database
- Refreshing a Partition
- Retracting a Partition
- Previewing Partitions Prior to Export from The Master Database

Recovering an Obsolete Partition

To recover an obsolete partition, follow these steps:

1. Ensure that the partition to be recovered has a status of *Exported*.
2. Rename or move the partition file to be recovered to prevent the export process from overwriting it.
3. Choose the partition file to recover. Update the path to the `.dpf`, `.dpm`, or `.dps` file, if necessary.
   The location display appears red to indicate that the file is obsolete.
4. Click *Import*. The status bar reports success or failure.
   a. If the recovery is successful, save the drawing.
   b. If not, import the file that you exported previously, or quit the drawing to return it to the previous state.

   ⚠ During recovery, no pin or gate swaps nor schedule updates are imported. If component assignments fail, then the symbol becomes unplaced.

Related Topics

- workflow
- Workflow Manager
- Importing a Partition into a Master Database
- Exporting a Partition from a Master Database
- Refreshing a Partition
- Retracting a Partition
- Previewing Partitions Prior to Export from The Master Database
- Appending Notes to a Master or Partition Database

world cancel

The `world cancel` command cancels the command currently being executed, similar to *Cancel* in the Design window frame. For more information on the *WorldView* window and commands related to it, ssee the *Getting Started with Physical Design* user guide in your documentation set.

world done

The `world done` command completes the command currently being executed, similar to *Done* in the Design window frame. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

world drawing2port

The `world drawing2import` command redisplays the full extent (left–to–right and top–to–bottom) of the drawing in the Design Window; that is, it zooms back so the entire drawing displays in the current Design Window frame. Use this command to view the entire drawing. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

world find next

The `world find next` command centers the drawing display about the next element highlighted in the World window. Elements displayed in the *WorldView* window by default (for example, router keepin or design outline) are not used for *Find Next* unless they are highlighted. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

world find previous

After executing *Find Next*, you can perform *Find Previous* to return to center on the previously displayed, highlighted element. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

world move

The `world move` command c hanges the position of the current window over the drawing. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

Changing the Position of the Current Window over the Design

To change the position of the current window over your deisgn, perform the following steps:

1. Run `world move`.
   The current window outline centers on the cursor, and moves dynamically within the *WorldView* window.
2. Click left to choose a new position for the current window.
   The content of the new window appears in the Design Window

world oops

The `world oops` command reverses the effect of the previous selection, similar to *Oops* in the Design window frame. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

world resize

The `world resize` command zooms in the display on a new window area that you choose by clicking points in the *WorldView* window. For more information on the *WorldView* window and commands related to it, see the *Getting Started with Physical Design* user guide in your documentation set.

Zooming into Current Window over the Design

To zoom into the current design over the design, perform the following steps:

1. Run `world resize`. The *WorldView* window prompts you to click the first corner of a new window.
2. Click the first corner. The program displays a dynamic rectangle anchored at that point.
3. Click the second corner of the rectangle. The program repaints the new window area in the Design Window.

write

The `write` command writes the currently active layout with the name <*filename*>, while keeping the layout displayed and active. The difference between the `write` command and the `save` command is that `write` does not change the name of the active layout.

The layout editor displays a browser form prompting for Layout name *abc*, where *abc* is the name of the currently active layout. It writes the layout to this name if you click *OK* without typing a different name into the *File Name* field.

Syntax

```
write <filename>
```

Saving the layout to a file with a different name

Follow these steps to save the layout to a file with a different name:

1. Enter the new name into the *File Name* field, and click *OK*.
   If layout <*filename*> already exists, a confirmer window displays:

   ALLEGRO LAYOUT: /<*path*>/<*filename*>: File Exists. Overwrite?

2. Click *Yes* to proceed or *No* to cancel the write.
   Regardless of the name written to, the active layout name does not change.