# PADS Layout Advanced Packaging Tutorial

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

# Table of Contents


Creating a Die Package ........ 1
Creating Ball Grid Array Patterns ........ 6
Creating a Substrate for the Package ........ 10
Defining Layers and Design Rules ........ 11
Creating a Wire Bond Fanout ........ 16
Editing Wire Bond Pads ........ 21
Connecting With a Netlist ........ 25
Connecting Without a Netlist ........ 28
Connecting With the Route Wizard ........ 39
Adding Teardrops ........ 43
Creating Die Flags and Power Rings ........ 45
Connecting Power and Ground Pads ........ 47
Creating Copper Areas ........ 50
Creating a Wire Bond Diagram ........ 52
Creating Wire Bond Reports ........ 55

# Creating a Die Package

In this lesson, you'll use the Die Wizard to parametrically construct a die part definition. You will also learn how to import die pad data from a text file and modify the imported data.

### In this lesson:

- Entering parametric die data
- Importing die data from an ASCII text file
- Modifying imported die data
- Adding the die to the design
- Understanding the ASCII text file format

### Restriction

- This tutorial requires the General Editing licensing option. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **previewrules.pcb** in the \PADS Projects\Samples folder.

Use a Start-up file to set preferences required for this tutorial.

## Open a start up file

**File menu > New**

### To use the Start-up file:

1. In the Set Start-up File dialog box, select **PBGAtutorial.stp**.

   Tip: If the Set Start-up File dialog box doesn't appear, on the File menu, click Set Start-up File.

2. Click **OK**.

## Entering parametric die data

**BGA button > Die Wizard button**

You can use parametric construction to construct a die part definition in the absence of electronic data like GDSII or ASCII. Parametric construction is ideal for scenarios requiring package feasibility studies before the die is complete.

The Die Wizard provides a preview window that dynamically updates as you enter parameters.

### Defining the die outline

#### To define the outline size:

1. In the Create Die dialog box, click the **Parametrically** button.
2. In the Die Wizard - Create Parametrically dialog box click the **Die Size** tab.
3. In the Units area select **Metric**.
4. In the Die Part Type box, type **DIE248** as the name.
5. In the Length box, type **8** to specify a die length of 8 mm.
6. In the Width box, type **8** to specify a die width of 8 mm.

### Defining the chip bond pads

**Die Wizard – Create Parametrically dialog box > CBP tab**

#### To define the pads:

1. In the Total box, type **248** to specify a total pad count of 248.
2. Make sure that GND **%** and PWR **%** are set to **10**. This specifies 10% of the pads on a per-side basis to be GND and PWR.
3. In the Pad Pitch box, type **.12** to specify a pad pitch of .12 mm.
4. In the Distance from Die Edge box, type **.1**.
5. In the Pad Shape area, on the Shape list, select **Rectangle**.
6. In the Length box, type **.07** to specify a pad length of .07 mm.
7. In the Width box, type **.07** to specify a pad width of .07 mm.

Tip: Select any of the cells in the spreadsheet to override total pad, GND, or PWR pad count values.

### Defining the pad numbering

**Die Wizard – Create Parametrically dialog box > Pad # tab**

#### To define the pad numbering:

- In the Pad 1 Side area, select **Top** to move pin 1 to the left side of the top row of pads.

### Defining pad function names

**Die Wizard – Create Parametrically dialog box > Pad Functions tab**

#### To define the pad function names:

- Although no changes are required in this area, browse through the pad list to review the pad function names.

Tip: Select any of the pad function name cells to override pad function names.

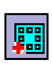
### Defining die wizard preferences

**Die Wizard – Create Parametrically dialog box > Die Prefs tab**

#### To define the die wizard preferences:

1. In the Part Creation Mode area, click **Add Part to Design**. This automatically adds the die part to the design.
2. You have defined the die part. For the purpose of this exercise, click **Cancel** since you are importing an ASCII file in the next section. If you were to click **OK**, the part definition would be created automatically and added to the design.

## Importing die data from an ASCII text file

**Die Wizard button** 

#### To import the data:

1. In the Create Die dialog box, click the **From Text File** button, to import die data from an ASCII text file.
2. In the Die Wizard - Create from Text File dialog box, click **Browse**.
3. Navigate to the \PADS Projects\Samples folder and click **Die248.csv**.
4. Click **Open** to import the contents of the file into the Die Wizard.

   Result: The Preview window displays the imported die data.

## Modifying imported die data

**Die Wizard – Create from Text File dialog box > Die Size tab**

Once you import the die data into the Die Wizard, you can modify or override most aspects of the data: die outline, origin, pad numbering, pad size, pad shape, and pad function names.

The default die outline is a bounding box surrounding the die pads. It often needs to be modified.

### To change the die outline:

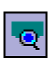
1. Make sure **Metric** is selected in the Units area.
2. In the Length box, type **8** to specify a die length of 8 mm.
3. In the Width box, type **8** to specify a die width of 8 mm.

## Adding the die to the design

**Die Wizard – Create from Text File dialog box > Die Prefs tab**

### To add the die part to the design:

1. In the Part Creation Mode area, click **Add Part to Design**. This adds the part die to the design automatically when you close the dialog box.
2. Click **OK**.



3. On the standard toolbar, click the **Board** button to resize the window view.

## Understanding the ASCII text file format

In the previous exercise you imported an ASCII text file into the Die Wizard to initiate the Die Part definition. This data may come from the IC place and route design system, a spreadsheet program like Microsoft® Excel, or a text editor.

### File format

The first line of the file must specify the units. Acceptable values are, **Mil**, **MM**, **Micron**, and **Inch**. These values are not case sensitive.

The pad data must be in the following format to import correctly into the Die Wizard. A comma must separate each field.

### Pad data format:

| Pad# | Pad Function | Xcoord | Ycoord | Pad Length | Pad Width |
|---|---|---|---|---|---|
| 1 | GND | -3.66 | 3.865 | 0.07 | 0.07 |
| 2 | PWR | -3.54 | 3.865 | 0.07 | 0.07 |
| 3 | SIG003 | -3.42 | 3.865 | 0.07 | 0.07 |

### Pad data explanation:

| Data | Description |
|---|---|
| Pad# | The chip bond pad number |
| Pad Function | The pad function name |
| Xcoord | The X distance from the origin of the die |
| Ycoord | The Y distance from the origin of the die |
| Pad Length | (Optional) The length of the die bond pad. When no value is specified one is derived automatically. |
| Pad Width | (Optional) The width of the die bond pad. When no value is specified, one is derived automatically. If the Pad Length is specified but the Pad Width is not specified, the pad is assumed to be circular. |

### Sample Text File:

```
MM,,,,,
1,GND,-3.66, 3.865, 0.07, 0.07
2,PWR,-3.54, 3.865, 0.07, 0.07
3,SIG003,-3.42, 3.865, 0.07, 0.07
4,SIG004,-3.3, 3.865, 0.07, 0.07
5,SIG005,-3.18, 3.865, 0.07, 0.07
```

- Do not save a copy of the design.

**You completed the creating a die package tutorial.**

# Creating Ball Grid Array Patterns

In this lesson, you'll learn how to create a ball grid array (BGA) and add it to the design.

Component footprints are called PCB decals in PADS Layout. To create a BGA you must use the Decal Editor to create the pattern as a PCB decal, create a part type, and assign the BGA decal to the part type.

### In this lesson:

- Creating a BGA decal
- Saving the decal
- Adding the BGA component to the design

### Restriction

- This tutorial requires the ECO, Library Editor and General Editing licensing options. On the Help menu, click **Installed Options** to determine whether you can proceed.

Requirement: You must understand the concept of PADS Layout library parts before proceeding with this tutorial. See the section called *Understanding the PADS Layout part type and PCB decal* in the *Creating library parts tutorial*.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_1.pcb** in the \PADS Projects\Samples folder.

## Creating a BGA decal

The Decal Editor provides a BGA Pin Wizard. In this wizard you enter specific parameters for the BGA and the wizard automatically creates the BGA footprint. In this tutorial, you will create a BGA package with 23 rows and 23 columns with a 1.27mm pitch and .75mm ball pad.

## Using the BGA pin wizard

**Tools menu > Decal Editor > Drafting button > Wizard button**

Use the Pin Wizards dialog box to create complex pad array patterns after you specify the pattern.

### To create the BGA footprint:

1. In the Pin Wizards dialog box click the **BGA/PGA** tab.
2. In the Units area, select **Metric**.
3. In the Origin area, select **Center**.
4. In the Decal Type area, select **Substrate**. The message *Substrate Decal Type selected, terminal numbering will be mirrored* appears. Click **OK**.
5. In the Silk Screen area, clear **Create**.
6. In the Pad Stack area, click **SMD**.
7. In the Diameter box, type **.75**.
8. Select **Assign JEDEC Pinning**.
9. In the Row Pitch and Column Pitch boxes, type **1.27**.
10. In the Row Count and Column Count boxes, type **23**.
11. In the Void Rows and Void Columns boxes, type **15**.
12. In the Center Rows and Center Columns boxes, type **5**.
13. Click **OK**. The completed PCB decal appears in the workspace.

### Substrate or component decal type

When creating a BGA pattern for a component (a BGA to mount on the PCB design), you create a BGA with pin A1 in the upper left corner of the pattern. However, when creating BGA patterns to add to the design of a BGA substrate, you orient the decal with pin A1 in the upper right corner of the pattern because the BGA pattern is mirrored and placed on the bottom layer of the design. The final position of the BGA pattern has pin A1 in the upper left.

## Saving the decal

**File > Save Decal**

Before you can add the BGA to your design, save the decal and create a part type. In PADS Layout these two steps are combined into a single operation.

### To save the decal to the library:

1. On the Save Alphanumeric to Library dialog box, in the **Library** list, select **\Program Files\Mentor Graphics\PADS\ *<latest_release>*\Libraries\usr**.
2. In the Name of PCB Decal box, type **BGA329.**
3. Select **Save Alphanumerics** to enable other parts of the dialog box.
4. Select **Create New Part Type**.
5. In the Part Type area, in the **Library** list, select **\Program Files\Mentor Graphics\PADS\ *<latest_release>*\Libraries\usr.**
6. In the Name of Part Type box, type **BGA329**.
7. Click **OK** to close the dialog box and save the decal and part type. If a prompt appears asking to confirm overwriting an existing part decal or part type, click **Yes**.
8. On the **File** menu, click **Exit Decal Editor** to leave the Decal Editor and return to the Layout Editor.

## Adding the BGA component to the design

**BGA button**  **> Add Component button** 

You can now add the BGA component to the design and place it on the bottom layer.

### To add the BGA from the Layout Editor:

1. In the Get Part Type from Library dialog box, in the Items box, type **bga*** and click **Apply**.

   Result: The preview:BGA329 part appears in the Part Types list.
2. Select **usr:BGA329** and click **Add** to add an instance of the part type to the design.

   Result: A dialog box appears prompting you to enter a prefix for the part to add. This is the prefix of the part's reference designation (for example, R for resistors or C for capacitors).
3. Type **BGA** in the box and click **OK**.

Result: An instance of the BGA329 part type, named BGA1, is attached to the pointer.

4. Click **Close** on the Get Part Type from Library dialog box.
5. To place the BGA pads on the bottom side of a BGA substrate, right-click and click **Flip Side**. This mirrors the part and places it on the bottom layer of the design.
6. Position your pointer over the design origin marker at the center of the die component and click the left mouse button to place the BGA component at the design origin.

   Tip: You can also use the Search modeless command to assist in placing the part. Type **s 0 0**, press **Enter**, and then press **Spacebar**.

7. On the standard toolbar, click the **Board** button to resize the window view.
8. Do not save a copy of the design.

**You completed the creating ball grid array patterns tutorial.**

# Creating a Substrate for the Package

The next step in designing advanced packages with PADS Layout is to create a substrate outline, also known as a *board outline*.

### In this lesson:

- Creating a board outline

### Restriction

- This tutorial requires the Drafting Editing and Physical Design Reuse licensing options. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_2.pcb** in the \PADS Projects\Samples folder.

## Creating a board outline

A board outline is created using the same polygon creation methods used to create drafting items.

1. Drafting button > Board Outline and Cut Out button .
2. Type **g1** and then press **Enter** to set all grids to 1 mm.

   Result: The message *All grids set to 1 1* appears in the status bar.
3. Type **gd1** and then press **Enter** to set the Display Grid to 1.

   Result: The message *Dot grid set to 1 1* appears in the status bar.

   Tip: You may have to zoom in to see the grid.
4. Right-click and click **Rectangle**.
5. Move the pointer to location -15,-15 and click the left mouse button. You can locate -15,-15 using the Search modeless command. Type **s -15 -15** and press **Enter**. A dynamic rectangle attaches to the pointer.
6. Move the pointer to 15, 15 and click the left mouse button to complete the board outline. You can use the Search modeless command here as well.
7. On the standard toolbar, click the **Board** button to center the board in view.
8. Do not save a copy of the design.

**You completed the creating a substrate for the package tutorial.**

# Defining Layers and Design Rules

Design rules include clearance, routing, and high-speed constraints assigned as default conditions or assigned for nets, layers, class groups, or pin pairs. You can also assign conditional design rules and differential pairs.

### In this lesson:

- Setting the layer arrangement for the BGA substrate
- Setting layer stackup
- Modifying the default via
- Setting default clearance rules
- Using on-line design rule checking (DRC)
- Setting display colors

### Restriction

- This tutorial requires the ECO, General Editing and Physical Design Reuse licensing options. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_3.pcb** in the \PADS Projects\Samples folder.

## Setting the layer arrangement for the BGA substrate

**Setup menu > Layer Definition**

With PADS Layout you can define the layer arrangements for the BGA design. This includes assigning the number of layers, nets associated with embedded plane layers, layer stackup, and layer thickness.

The tutorial design will be fabricated as a two-layer substrate.

### To set the first layer:

1. In the Layers Setup dialog box, select **Top** in the list of layers.
2. In the Name box type **Die Side**.
3. In the Electrical Layer Type area, click **Component**.
4. In the Plane Type area, click **No Plane**.

5. Set the Routing Direction to **Vertical**.

### To set the second layer:

1. Select **Bottom** in the list of layers.
2. In the Name box type **BGA Side**.
3. In the Electrical Layer Type area, click **Component**.
4. In the Plane Type area, click **No Plane**.
5. Set the Routing Direction to **Horizontal**.

Tip: Do not close the Layers Setup dialog box.

## Setting layer stackup

**Layers Setup dialog box > Thickness button**

A typical layer stackup for a two-layer FR4 substrate is composed of a fiberglass substrate copper clad on both sides. Use the Layer Thickness dialog box to set the layer stackup values.

1. In the Layer Thickness dialog box on the line for the Die Side layer, double-click the **Thickness** cell. The cell switches to in-place editing mode.
2. In the Thickness cell type **.01** (mm).
3. On the line for the BGA Side layer, double-click the **Thickness** cell. The cell switches to in-place editing mode.
4. In the Thickness cell type **.01** (mm).
5. On the line for the Substrate type double-click the **Thickness** cell. Type **2** (mm) to set thickness. On the same line, double-click the **Dielectric** cell, and then type **4.5** to set dielectric constant value.
6. Click **OK** to close the Layer Thickness dialog box.
7. Click **OK** to close the Layers Setup dialog box.

## Modifying the default via

**Setup menu > Pad Stacks**

As preparation for route editing, you need to modify the default via definition.

### To modify the default via:

1. In the Pad Stacks Properties dialog box, in the Pad Stack Type area, click **Via**.
2. Select **STANDARDVIA** in the Decal Name list.
3. In the Sh.: Sz.: Layer: (shape, size, layer) list, select **CNN 0.254 <Start>**.

4. In the Diameter box type **.4**.
5. In the Sh.: Sz.: Layer: list select **CNN0.254 <Inner Layers>**.
6. In the Diameter box, type **.4**.
7. In the Sh.: Sz.: Layer: list select **CNN 0.254 <End>**.
8. In the Diameter box, type **.4**.
9. In the Drill Size box, type **.16**.
10. Click **OK** to save the via definition and close the Pad Stacks Properties dialog box. The message *Are you sure you want to change all vias of type STANDARDVIA* appears.
11. Click **Yes** to change all vias of type STANDARDVIA.

## Setting default clearance rules

**Setup menu > Design Rules > Rules dialog box > Default button > Default rules dialog box > Clearance button**

With PADS Layout, you can define constraints for clearance and routing, and high-speed constraints for each layer of the design rule hierarchy. The Clearance area of the Clearance Rules dialog box contains a matrix of design data. The matrix data lets you specify clearance values for each or all data types.

### To assign clearance rules:

1. Set a global default clearance value by clicking **All** in the upper left corner of the matrix.
2. In the Input Clearance Value dialog box type **.07** and then click **OK**.

   Result: All matrix values change simultaneously.
3. In the Trace Width area, type:
   - **.07** in the Minimum box
   - **.07** in the Recommended box
   - **.5** in the Maximum box.
4. In the Same Net area, type **.07** in the following boxes:
   - Via to Via
   - SMD to Corner
   - SMD to Via
   - Pad to Corner.
5. In the Other area, type **.07** in the Drill to Drill and Body to Body boxes.
6. Click **OK** in the Clearance Rules dialog box.
7. Click **Close** in the Default Rules dialog box.

8. To save the changes, click **Close** in the Rules dialog box.

## Using on-line design rule checking (DRC)

You can enable real-time design rule checking during placement and routing operations to ensure that design constraints are maintained throughout the design process. This interactive checking is called DRC. You set DRC modes in the Preferences dialog box or by using the dr modeless commands.

There are four modes of DRC operation:

| DRC Mode | Description |
|---|---|
| DRC Off | Turns off design rule checking.<br>Modeless command: dro. |
| DRC Ign Clr | Ignores all clearance rules but prevents the intersection of traces during routing.<br>Modeless command: dri. |
| DRC Warn | Generates error messages for violations.<br>Modeless command: drw. |
| DRC Prevent | Prevents you from creating violations.<br>Modeless command: drp. |

## Setting display colors

**Setup Menu > Display Colors**

You assign or change layer colors and make individual items visible or invisible in the Display Colors Setup dialog box. You can also set the color for the screen background, board outline, and other elements.

### Assigning a new color to the BGA

1. Click light blue in the Selected Color area.
2. In the Color by Layer area, select each color box for the items under the Design Items area in the row associated with BGA Side to assign the light blue color.

### Assigning colors to other items

1. Click yellow in the Selected Color area.
2. In the Color by Layer area, select the **Errors** color box under Design Items for the Die Side and BGA Side layers to assign the yellow color for errors.

### Saving the color arrangement

You can save color arrangements to reuse from design to design. When you finish assigning colors to items in the Display Colors Setup dialog box, save the color arrangement.

1. Click **Save**.
2. In the Save configuration dialog box, type **All Items Visible**.
3. Click **OK** to save the configuration.
4. If the new configuration name does not appear in the **Configuration** area, select it from the list.
5. Click **OK** to apply the colors and to close the Display Colors Setup dialog box.
6. Do not save a copy of the design.

**You completed the defining layers and design rules tutorial.**

# Creating a Wire Bond Fanout

In this lesson, you'll create the wire bond fanout pattern.

### In this lesson:

- Creating a wire bond fanout
- Exploring additional wire bond options

### Restriction

- There are no restrictions for this tutorial.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_4.pcb** in the \PADS Projects\Samples folder.

## Creating a wire bond fanout

**BGA button**

### To start the Wire Bond Wizard:

1. Right-click and click **Select Components**.
2. Select die part **U1**. Use the Tab key if necessary to cycle through the selections.
3. On the BGA toolbar, click the **Wire Bond Wizard** button to open the Wire Bond Wizard for U1.

## Defining the ring geometry

**Wire Bond Wizard dialog box > Guide tab**

### To define the ring sizes:

1. In the SBP Rings area of the Wire Bond Wizard, select **Ground**.
2. In the Size area, type **.5** in the **X** and **Y** boxes, and then click **Distance from Die**.
3. In the SBP Rings area, select **Power**.
4. In the Size area, type **1.25** in the **X** and **Y** boxes, and then click **Distance from Die**.

5. In the SBP Rings area, select **Signal**.
6. In the Size area, type **2** in the **X** and **Y** boxes, and then click **Distance from Die**.
7. In the Shape area, select **Arced** in the list, and then type **.8** into the Height box.

## Setting fanout preferences

**Wire Bond Wizard dialog box > Fanout Prefs tab**

### To set ring fanout preferences:

1. In the SBP Rings area, Ctrl+click both the **Ground** and **Power** rings.
2. In the Substrate Bond Pad area, in the Shape list, select **Rectangle**.
3. In the Length and Width boxes, type **.04**.
4. In the Focus area, select **CBP** to specify substrate bond pad alignment to the component bond pads.
5. In the Wire Bond area, type **.025** in the Width box, and then type **0** in the Offset box.
6. In the SBP Ring area select the **Signal** ring.
7. In the Substrate Bond Pad area, in the Shape list, select **Oval**.
8. In the Length box type **.3**.
9. In the Width box type **.1**.
10. In the Focus area, select **CBP** to specify substrate bond pad alignment to the component bond pads.
11. In the Wire Bond area, type **.025** in the Width box, and then type **0** in the Offset box.

## Assigning wire bond rules

**Wire Bond Wizard dialog box > Rules area > Wire Bond Rules button**

### To assign fanout rules:

1. Select all the rule check boxes to enable all checks.
2. In the WB to WB Clearance box type **.03**.
3. In the WB to SBP Clearance box type **.03**.
4. In the Min Length box type **.5**.
5. In the Max Length box type **2.9**.
6. In the Max Angle box, type **45**.
7. Click **OK** to accept the rules and close the Wire Bond Rules dialog box.

## Assigning chip bond pads to rings

**Wire Bond Wizard dialog box > Assign CBPs button**

### To assign pads to rings:

1. On the Assign CBPs to Rings dialog box, in the View by list, select **CBP Function**.
2. In the CBP Function column, select **GND**.
3. In the Assign to area select the **Ground** check box.
4. Click **Apply**.

   Result: All the CBPs with the GND function name will be assigned to the Ground ring.
5. In the CBP Function column, select **PWR**.
6. In the Assign to area, select the **Power** check box.
7. Click **Apply**.

   Result: All the CBPs with the PWR function name will be assigned to the Power ring.
8. Click the **Select All** button to select the remaining CBP function names.
9. In the Assign to area select the **Signal** check box.
10. Click **Apply**.

    Result: The remaining CBPs will be assigned to the Signal ring.
11. Click **Close** to close the dialog box.

## Specifying the wire bond fanout strategy

**Wire Bond Wizard dialog box > Strategy tab**

The Strategy tab contains options you can specify to improve routing and manufacturability.

### To specify the strategy:

1. In the SBP Rings area, Ctrl+click the **Ground** and **Power** rings.
2. In the Preferred Spacing area, select the **WB to SBP** check box.
3. In the WB to SBP box, type **.07** to specify the preferred wire bond to substrate bond pad spacing.
4. Select the **Force Preferred Spacing** check box.
5. In the SBPs Having Same Function Name area, select **Create Nets from Pin Function**.
6. In the SBP Rings area, select the **Signal** ring.
7. In the Preferred Spacing area, select the **SBP to SBP** check box.

8. In the SBP to SBP box, type **.1** to specify the preferred substrate bond pad to substrate bond pad spacing.
9. Select the **Force Preferred Spacing** check box.
10. In the SBPs Having Same Function Name area, select the **Create Nets from Pin Function** check box.

## Previewing and checking the wire bond fanout

### To preview the fanout:

1. In the **Preview Options** area of the Wire Bond Wizard dialog box, select the **Report** check box.
2. Click the **Preview Fanout** button. The wire bond fanout is displayed in the workspace and the Wire Bond Wizard Report is generated.
3. Browse the report. Notice there are Max Length violations on the Signal ring. These violations were introduced on purpose to illustrate rule checking and reporting.
4. Close the report.
5. Click the **Wire Bond Rules** button. Change the **Max Length** to **3,** then close the Wire Bond Rules dialog box. Refer to the "Specifying Wire Bond Rules" section, for more information.
6. Click the **Preview Fanout** button again. Browse the Wire Bond Wizard Report and notice the Max Length violations have been resolved.
7. Close the report.

## Generating the wire bond fanout

**Wire Bond Wizard dialog box > Create Fanout button**

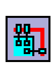
After you click the Create Fanout button, a dialog box appears indicating that GND and PWR nets are being created.

### To complete the fanout:

1. Click **Close** to close the dialog box.

   Result: The Wire Bond Wizard Report appears.
2. Browse the report. Close it when you are done.
3. Click **OK** to close the Wire Bond Wizard.

# Exploring additional wire bond options

**Wire Bond Wizard button** 

The previous steps covered the basics of creating the wire bond fanout patterns. You can use additional controls for fine-tuning or producing alternative results.

## Importing substrate bond pad function names from a netlist

The Substrate Bond Pad Function Name matches the Chip Bond Pad Function Name. In some cases you may want the Substrate Bond Pad Function Name to reflect the associated net name. SBP Properties enable the netlist to scan to derive the Substrate Bond Pad Function Name from the net names.

### To import pad function names:

1. Click the die in the workspace. This opens the Wire Bond Wizard for U1.
2. In the Wire Bond Wizard dialog box, click the **SBP Properties** button.

   The Substrate Bond Pad Function Name matches the Chip Bond Pad Function Name.
3. Click the **Derive from Netlist** button.
4. Navigate to the \PADS Projects\Samples folder, and then double-click the **SBPnetlist.asc** file to open it.
5. Select **U1** in the Select Component area and then click **OK**.

   Result: The SBP function names now reflect their associated net name.
6. Click **OK** to close the SBP Properties dialog box.

## Saving and loading setup files

**Wire Bond Wizard dialog box > Load Setup button**

When you are satisfied with the wire bond setup, you can save the parameters to a file for use on other designs. Clicking Save Setup writes all the parameters to a file. Chip Bond Pad assignments are not written because they differ from design to design.

### To load a pre-existing wire bond setup file:

1. Navigate to the \PADS Projects\Samples folder, and then click the **PBGAtutorial_4.wbw** file.
2. Click **Open** to read the file.
3. Do not save a copy of the design.

**You completed the creating a wire bond fanout tutorial.**

# Editing Wire Bond Pads

Once you create a wire bond fanout pattern in PADS Layout you may want to make some adjustments.

### In this lesson:

- Modifying substrate bond pads
- Checking wire bond rules
- Adding wire bonds and substrate bond pads

### Restriction

- This tutorial requires the General Editing and Verify Design licensing options. Click **Installed Options** on the Help menu to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_5.pcb** in the \PADS Projects\Samples folder.

## Modifying substrate bond pads

### Changing bond pad shapes

#### To change the shape of a substrate bond pad:

1. Zoom into the upper right portion of the wire bond fanout.
2. Use the modeless command **g .01** to specify a .01 grid.
3. Right-click and click **Select Traces/Pins**.
4. Using the modeless Search command, type **s u1.65** and then press **Enter**. This moves the pointer to the U1.65 substrate bond pad. Press **Spacebar** to select it.
5. With the U1.65 substrate bond pad selected, right-click and click **Query/Modify SBP**.
6. In the Shape area, in the Shape list, select **Rectangle**.
7. In the Shape area, type **.5** into the Length box. Click **Apply**.

   Result: The substrate bond pad has been updated with the new parameters.
8. Click **Cancel** to close the dialog box.

9. On the standard toolbar, click the **Undo** button to set the substrate bond pad back to its original parameters.

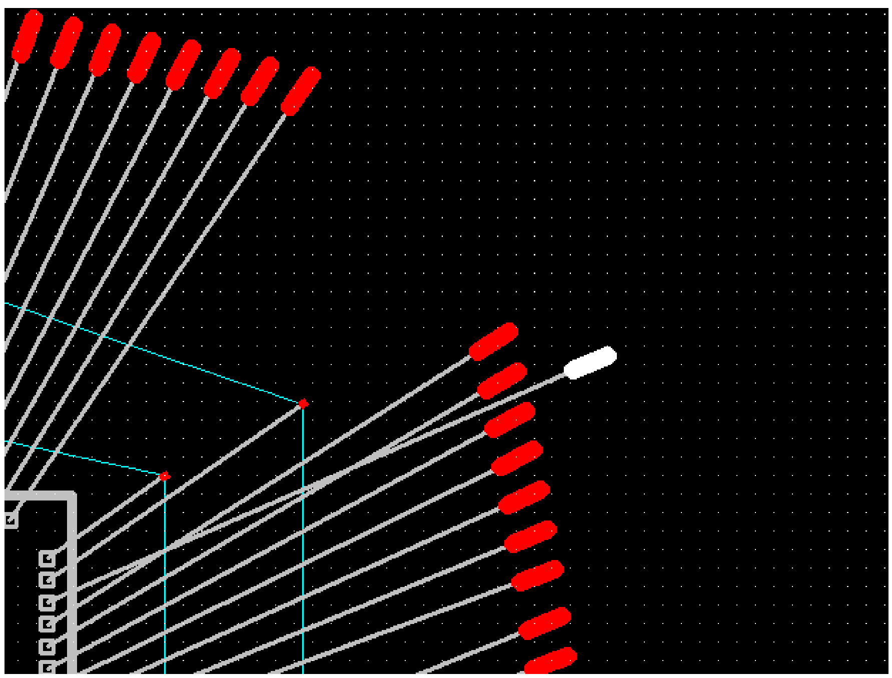
Tip: You can use the Query/Modify SBP dialog box to modify the parameters for an individual or a group of substrate bond pads.

## Moving substrate bond pads

### To move a bond pad:

1. Select substrate bond pad **U1.65**.
2. Right-click and click **Move SBP**.
3. Move your pointer and notice how the substrate bond pad is attached dynamically. Also notice the rings defined in the Wire Bond Wizard are now displayed for reference.

### SBP and rings:



4. Move the pointer so the substrate bond pad is off the signal ring. Now move the substrate bond pad back onto the signal ring and notice the automatic snapping.
5. Move and place the substrate bond pad as shown.

Tip: You can set the parameters for wire bond snapping on the Die Component tab of the Preferences dialog box.

## Checking wire bond rules

**Tools menu > Verify Design**

### To check the wire bonds:

1. Select **Wire Bonds** in the Check area.
2. Click **Start**. The message *Wire Bond rules checking has been done for the current window. Number of errors found- ?* (2 or 3, depending upon the placement) appears.
3. Click **OK** in the message box.
4. Notice the violations that appear in the Location area and the reason for the violations in the Explanation area. Also notice the error markers in the area of the design where the recent wire bond edits were performed. Review the violations.
5. Click the **Clear Errors** button. Click **Close** to close the dialog box.

6. On the standard toolbar, click the **Undo** button to reset the substrate bond pad back to its original location.

## Adding wire bonds and substrate bond pads

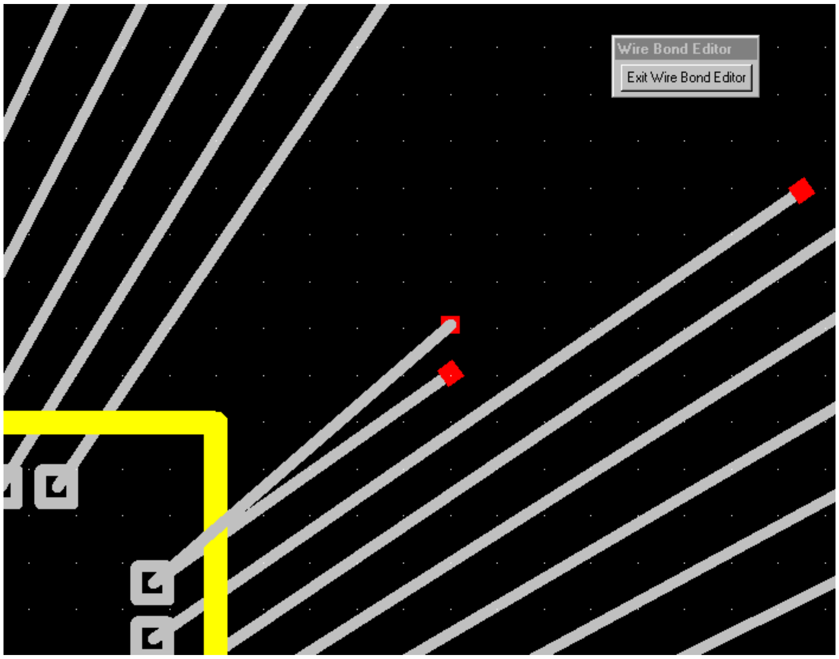
**BGA button**  **> Wire Bond Editor button** 

Some nets may require additional wire bonds and substrate bonds to increase current capacity.

### To add a pad:

1. Select the **U1** die component.
2. Zoom into the upper right portion of the wire bond fanout.
3. Right-click and click **Add SBP**.
4. In the Add Substrate Bond Pad dialog box, type **GND** in the Function box.
5. Click **Add** to accept the parameters and close the dialog box.
6. A new substrate bond pad attaches to the pointer and the rings appear for reference.
7. Place the new substrate bond pad above the upper-most substrate bond pad on the right side of inner ring (GND). The new substrate bond pad remains selected.
8. Right-click and click **Add WB**. A new wire bond appears on the pointer originating from the new substrate bond pad.

### SBP placement and wire bond connection:




9. Connect the new wire bond to the upper-most chip bond pad on the right side of the die.
10. Right-click and click **Cancel**.
11. Click **Exit Wire Bond Editor**.

    Tip: After making interactive wire bond edits, you should use Verify Design to check the edits.

12. Do not save a copy of the design.

## You completed the editing wire bond pads tutorial.

# Connecting With a Netlist

You can establish a netlist and create connections interactively using tools on the BGA toolbar.

### In this lesson:

- Importing a partial netlist
- Displaying and hiding connections
- Creating connections interactively
- Swapping pins

### Restriction

- This tutorial requires the ECO and Verify Design security options. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_5.pcb** in the \PADS Projects\Samples folder.

## Importing a partial netlist

**File menu > Import**

In this activity you assign power and ground to the ball pads.

1. Navigate to the \PADS Projects\Samples folder and select the file **PGnetlist.asc**.
2. Click **Open** to import the netlist.

In this file, PWR and GND are assigned to their respective ball pads.

## Displaying and hiding connections

**View menu > Nets**

For a less cluttered display you can turn connections on or off. In this step you undisplay the power and ground connections but leave the pins highlighted.

1. In the **Net List** area, select both the **PWR** and **GND** nets.
2. Click **Add** to add PWR and GND to the view list.
3. In the View List area, select both the **PWR** and **GND** nets.

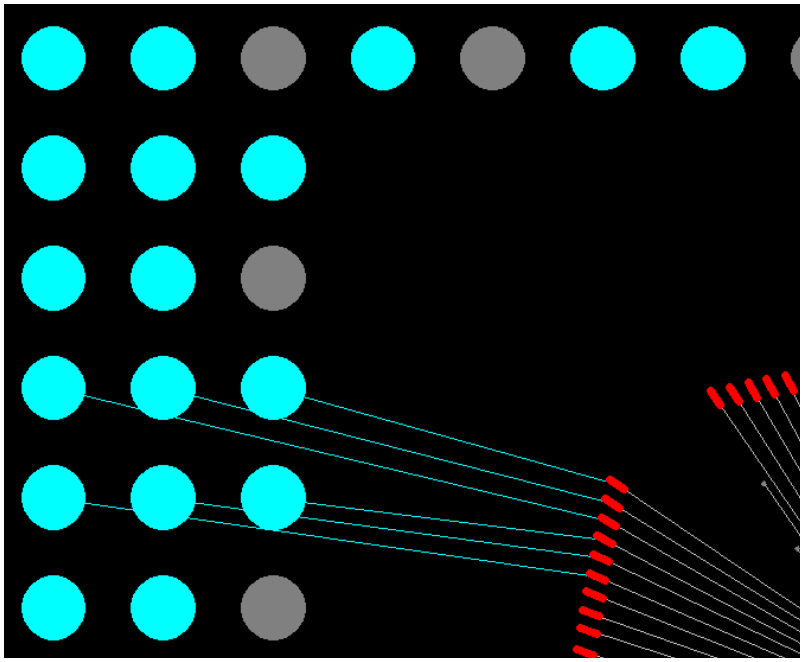
4. In the View Unroutes Details area, select **None**.
5. In the Color by Net area, select the dark gray color box to highlight the PWR and GND pins.
6. Click **OK** to accept the changes and close the dialog box. The connections are removed from the display but the PWR and GND pins are highlighted.

## Creating connections interactively

**BGA button > Add Connection button**

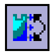
Some advanced IC package designs do not require a fixed netlist. In these cases the connectivity is left to the designer's discretion. The following exercises show you how to create connections interactively and swap pin assignments.

### Zoom area:



1. Zoom into the upper left area of the wire bond fanout pattern as shown.
2. Using the modeless Search command, type **s u1.248** and then press **Enter**. Press **Spacebar** to start the connection.
3. Right-click and click **Derive Net Name from Pin Function**. This assigns the net name automatically based on the pin function name.
4. Using the modeless Search command type **s bga1.g4** and **Enter**. Press **Spacebar** to select the pin.
5. Press **Esc** to end the connection.
6. Repeat these steps to generate the connections shown above.

## Swapping pins

**Swap Pin button** 

1. Type the modeless Search command **s bga1.g4** and then press **Enter**.
2. Press **Spacebar** to select the first pin to swap. The remaining pins will dim.
3. Select the ball pad to the immediate left (BGA1.G3)..
4. On the Confirm Pin Swap dialog box, select **Don't display again**.
5. Click **OK** to close the dialog box.

   Result: The connections have been swapped between the pins.
6. Repeat these steps for several more swaps. If you make a mistake, right-click and click **Undo Last Swap**.
7. Do not save a copy of the design.

**You completed the connecting with a netlist tutorial.**

# Connecting Without a Netlist

There are several tools in PADS Layout for interconnecting substrate bond pads to BGA pads. In this tutorial, you'll use several tools to connect the die component to the BGA.

### In this lesson:

- Using the manual route editor
- Starting routing
- Completing traces
- Deriving net names
- Guiding the pad entry
- Smoothing the pad entry and exit
- Modifying traces
- Using the Dynamic Route Editor
- Copying traces
- Creating a fanout pattern

### Restriction

- This tutorial requires the Dynamic Route Editing and General Editing security options. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_6.pcb** in the \PADS Projects\Samples folder.

## Using the manual route editor

The basic route editor is the core route editing function of PADS Layout. Many of the operations used to create traces in the route editor resemble other operations in PADS Layout, such as those used for creating polygons and line items. This minimizes your learning curve by letting you apply the same operations to many areas of PADS Layout.

Typically in PADS Layout, all connections are converted to traces by selecting the connection, digitizing new corners, and introducing layer changes using mouse and keyboard combinations. In PADS Layout you can also start interconnecting components without a netlist. In the following exercises, you'll use the route editor to create routes on the fly.

### Resizing the view

1. Press **Ctrl+B** to view the entire design.
2. Zoom into the upper-right part of the U1 component.
3. Locate substrate bond pad 65 using the Search modeless command. Type **s u1.65** and press **Enter**. The pointer moves to the location of U1.65.

### Setting a routing and via grid

For easier trace and via positioning, set a routing grid of 0 and a via grid of .1. To set the grids, use modeless commands:

1. Set the working grid to 0 by typing **g0** and pressing **Enter**. The message *All grids set to 0.00025 0.00025* appears in the status bar.
2. Set the via grid to .1 by typing **gv.1** and pressing **Enter**. The message *Via grid set to 0.1 0.1* appears in the status bar.

## Starting routing

**BGA button**  **> Add Route button** 

1. With nothing selected right-click and click **Select Pins/Vias/Tacks** to change the Selection Filter.
2. On the standard toolbar, select **Die Side** as the current layer from the Layer list.
3. Use the modeless command **ao** to set the angle mode to orthogonal.
4. Select substrate bond pad **U1.65** on the Die Side layer. The beginning of a new route segment dynamically attaches to the pointer.

   Tip: At this point in the tutorial, PADS Layout is in DRC Off mode. New trace segments are not prevented from shorting to other objects.

5. Once you initiate the new trace, move the pointer around and note how one end of the connection attaches to the component pin and the other end of the connection attaches to the pointer.

Tips:

- New segments are constrained to 90-degree increments from the origin of the trace segment. This is because the current trace angle mode is Orthogonal.

- At any point in this exercise you can exit the routing command operation by pressing the Esc key. You can also click the Undo button on the standard toolbar to undo any actions.

### Changing the trace angle mode

You can change the trace angle mode during routing by clicking commands on the shortcut menu.

#### To change the trace angle mode:

1. While the trace is attached to the pointer, right-click, point to **Angle Mode**, and click **Diagonal**.
2. Move the pointer around. Notice how new segments are now constrained to 45-degree increments.

### Adding and deleting corners

You can add new corners to a trace segment by clicking the left mouse button. To remove new corners, press the Backspace key. Experiment with inserting and deleting new trace corners.

### Changing layers

You can change layers while routing in the same way that you add corners, except Shift+click. You can change layers at the current pointer location or at the last corner location while routing.

#### To initiate a layer change at the current pointer location:

- With a new trace segment attached to the pointer, Shift+click.

  Result: A new via is added at the location of the click and the second layer of the routing layer pair is now the current layer.

  Alternative: To initiate a layer change at the last corner location, press F4 or right-click and click Layer Toggle while a new trace segment is attached to the pointer. Vias are not added if the last trace corner is located inside a component pad.

### Selecting a destination for new traces

Before you can complete a new trace added on the fly, you must select a destination. There are two methods to choose a destination for a new trace in PADS Layout: use the Select Target command or double-click the left mouse button.

#### To choose a destination with the Select Target command:

1. With a trace segment attached to the pointer, right-click and click **Select Target**.

2. Move the pointer over the destination target, and click.

   The pointer returns to the end of the trace segment.

### To choose a destination using a double-click:

- With a trace segment attached to the pointer, move the pointer over the destination target and double-click.

## Completing traces

You can complete a trace in two ways. Use the steps in the "Start Routing" section to route traces from the substrate bond pads.

Tip: Remember, you are routing between a substrate bond pad and the ball grid pad, which are on different layers. You must add a via to complete the trace.

### To complete a trace using the Complete command:

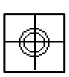
1. Start a new trace from one of the substrate bond pads.
2. Select a destination by right-clicking and clicking **Select Target**.
3. Shift+click to insert a via.
4. While the trace segment is attached to the pointer, right-click and click **Complete**.

   Alternative: Double-click your left mouse button.

   Result: The trace completes from its start to its destination and the trace pattern is "smoothed" or cleaned up.

### To complete a trace without using the Complete command:

1. Start a new trace from one of the substrate bond pads.
2. Select a destination by right-clicking and clicking **Select Target**.
3. Shift+click to insert a via.
4. With the trace segment attached to the pointer, define a trace pattern and position your pointer over the center of the BGA pad.



5. When the bull's-eye symbol appears, click.

   Result: The trace is completed, in many instances without "smoothing" or cleaning up the new trace.

Practice creating new traces on the fly using both target selection and completion methods.

Tip: At any point in this exercise you can exit the routing command operation by pressing the Esc key. You can also click the Undo button on the standard toolbar to undo any actions.

## Deriving net names

**Window menu > Status**

PADS Layout names all new nets with a system default of $$$<a number>. When designing for advanced packaging, you may prefer for the name of the net connecting the die pad to the BGA pad to match the function or the signal name of the die pad. For example, the die pad for RESET connects to a BGA pad and the name of the net that connects them is RESET.

PADS Layout can derive net names from pin function names when adding traces on the fly. You can enable and disable this option only on the shortcut menu that appears when the Add Route command is active.

### To enable this option:

1. Start a new trace from a substrate bond pad.

   Note that the net name appears at the top of the Status Window. This net name is a system default of $$$<a number> unless the net name is being derived from the pin function.

2. Immediately after starting the new trace, right-click and examine the **Derive Net Name from Pin Function** option. A check will appear to the left of the menu entry indicating that the option is enabled. If enabled, the net name appearing at the top of the Status Window matches the function name for the substrate bond pad.
3. Close the Status Window.

This option is considered a static option. It remains in its current state until you select the option again on the shortcut menu to change its state.

## Guiding the pad entry

**Tools menu > Options > Routing tab**

To achieve optimum pad entry and exit, you can guide pad entry and exit as you route. Use this option to manually digitize the path of the last segment entering or exiting a pad. With this option enabled, the trace angle temporarily switches to any angle when you start a trace. The trace angle remains in any angle mode until you place the first corner of the trace, then trace angle mode returns to the current setting.

### To enable Guide Pad Entry:

1. In the Pad Entry area, select the **Guide Pad Entry** check box.
2. Click **OK**.

**Result:** Routing behavior is different than before. The following exercises repeat some of the previous exercises using guide pad entry. Be sure to start on a pad that currently has no traces or connections.

## Starting routing

1. With nothing selected, right-click and click **Select Pins/Vias/Tacks**.
2. Set the current DRC mode to Prevent by typing the modeless command **drp** and pressing **Enter**.
3. Set the angle mode to orthogonal by typing the modeless command **ao** and pressing **Enter**.
4. Set the **Die Side** as the current layer by selecting it from the Layer list on the standard toolbar.

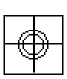
5. If not currently selected, click the **Add Route** button on the BGA toolbar.
6. Select the substrate bond pad for U1.65. As it did before, the beginning of a new trace segment attaches to the pointer.

### To demonstrate the new pad exit behavior:

1. Move the pointer in a circular fashion around the pad and notice how the first segment of the trace is an any-angle segment.
2. Move the pointer very close to the end of the pad and notice the octagonal DRC violation indicator. The indicator appears because placing a corner at the current pointer location would violate the same net SMD to Corner Clearance rule. Therefore, clicking to add a new corner is disabled until the pointer is positioned further away from the pad and the indicator disappears.
3. Move the pointer to a position where the indicator disappears, and then click to place the first corner.
4. Move the pointer in a circular fashion around the corner and notice how the segment of the trace is now an orthogonal segment.

### To demonstrate the new pad entry behavior:

1. Continue to route the new trace, right-click and click **Select Target** (rather than double-clicking) to select the target destination.



2. Add a few corners and then Shift+click to add a via. Move the pointer toward the center of the target pad.

   **Result:** The last segment immediately converts to an any angle segment when the bull's-eye symbol appears.

   **Tip:** You must be on the BGA side to see the bull's-eye symbol, which is why you add a via to change layers.
3. Click in the center of the target pad to complete the trace.

## Smoothing the pad entry and exit

**Tools menu > Options > Routing tab**

A second option for pad entry and exit determines whether or not the pad entry or exit is smoothed during routing. For this option, the pad entry or exit is defined as the first or last segment attached to the pad. When this option is disabled, the manually digitized pad entries and exits are locked and can't be adjusted during smoothing operations. By default, this option is disabled so that guided pad entries and exits are preserved.

#### To enable smooth pad entry and exit:

1. In the **Pad Entry** area select the **Smooth Pad Entry/Exit** check box.
2. Click **OK**.

### Experimenting with pad entry options

Perform the manual routing exercises above with the Smooth Pad Entry/Exit check box selected. This helps you become familiar with the operation of these options and how they affect route-editing behavior. When you are comfortable with the use and operation of the options, exit the current mode, clear both Pad Entry options on the Routing tab of the Preferences dialog box, and continue to the next topic.

## Modifying traces

You modify a trace by selecting a trace segment or via, and then using the commands available on the shortcut menu. You have more freedom to edit traces in DRC Off mode, but you can edit traces in any DRC mode.

### Deleting traces and trace segments

You can easily delete trace segments or pin pairs.

#### To delete traces or trace segments:

1. With nothing selected, right-click, and click **Select Anything**.
2. Select a segment of a completed trace and press **Delete**.

3. Click the **Undo** button on the standard toolbar to undo the deletion.
4. Shift+click on a trace (trace segment) to select the whole pin pair.
5. Press **Delete** to unroute the pin pair. If DRC is on at all (set to Prevent, Ignore, or Warn), the message *Delete Pin Pairs or Unroute Traces?* appears. Click **Unroute**.

### Experimenting with trace-editing commands

Experiment with the trace-editing commands by selecting various trace segments, vias, and corners. Use the shortcut menus or the keyboard shortcuts to initiate the move, stretch, split, add corner, add via, or other editing commands. See *PADS Layout Help* for more information about these commands.

## Using the Dynamic Route Editor

**BGA button > Dynamic Route button**

The Dynamic Route Editor (DRE) is another powerful interactive routing feature. Instead of digitizing each trace corner, as with the basic trace editor, you simply start the trace and move the pointer in the direction in which you want the trace to flow. Trace corners are dynamically added as you move the pointer.

#### To dynamically route:

1. With nothing selected, right-click and click **Select Pins/Vias/Tacks**. The Selection Filter updates.
2. Set the angle mode to Orthogonal by typing the modeless command **ao** and pressing **Enter**.
3. Set the DRC mode to Prevent by typing the modeless command **drp** and pressing **Enter**.
4. If necessary, zoom into the top of the design for easier routing.
5. Locate substrate bond pad 65 by typing the modeless search command **s u1.65** and pressing **Enter**.
6. Select pin 65 and start a trace using Dynamic Route. A trace dynamically attaches to the pointer.
7. Move the pointer up and to the left towards the substrate bond pads along the top of the die. Note how the trace stops at the edge of the pad even when you move the pointer over the adjacent bond pads.
8. Move the pointer until it is slightly above or below the adjacent bond pads. Notice how a new trace pattern is created around the obstacle automatically.

### Experimenting with DRE

Experiment with DRE by moving the pointer around obstacles with the trace attached. Once you are ready to select a destination, use the commands previously demonstrated in the manual routing exercises.

Tip: To back up a trace with DRE, slowly trace back over the newly created trace pattern.

1. Change the trace angle to Diagonal by typing **ad** and pressing **Enter**.
2. Continue experimenting. Try the same exercises with the trace angle set to Any Angle by typing **aa** and pressing **Enter**.

Tip: At any point in this exercise, you can exit dynamic route editing by pressing Esc. You can also click the Undo button on the standard toolbar to undo any actions.

### Route connections with DRE

Use DRE to complete traces. Many of the commands that apply to manual routing also apply to dynamic routing:

- The Backspace key removes the last corner added.
- Shift+click inserts a via at the current pointer location and causes a level change.
- Ctrl+click ends the trace at the current location with or without a via.

### Dynamic reroute with DRE

You can also use DRE to reroute traces the same way as the manual route editor.

#### To reroute a trace with DRE:

1. Right-click and click **Select Traces/Pins**.
2. Select any trace segment on any trace, right-click and click Dynamic Route if not already in Dynamic Route mode.
3. Create a new trace pattern and complete it by double-clicking on any other point in the trace or trace segment. Otherwise, use normal trace completion commands when completing component pins and vias during reroute.

   Tip: When you try to complete another point of the trace, the first click of the double-click attaches the new pattern to the existing pattern. The second click completes the reroute. It may take a few tries to adjust to completing traces this way. For best results, minimize mouse movements while double-clicking.

## Copying traces

You can duplicate traces or trace segments to speed repetitive tasks by copying and placing previously created traces. In this section, you'll use the trace copy features to quickly create fanouts from the BGA pads.

### Preparation

1. Open the file named **PBGAtutorial_6b.pcb** in the \PADS Projects\Samples folder.
2. On the View menu click **Board** to view the entire design.
3. Zoom into the upper right part of the design, centering the view on one of the BGA pads.

## Setting a few preferences

Ease the positioning of traces and vias by setting a routing grid and a few other preferences.

### To set the preferences:

1. Type **g0** and pressing **Enter**, to set the working grid to 0.

   Tip: Messages appear in the status bar as you complete these steps.

2. Type **gv.1** and press **Enter**, to set the via grid to 0.1.
3. Type **ad** and press **Enter**, to set the angle mode to diagonal.
4. Type **e** and press **Enter**, to set the end via mode.

   Result: The message *End Via Mode Set* appears in the status bar. If *End No Via Mode Set* appears instead, type e and press Enter until it appears.

5. Type **dro** and press **Enter**, to turn Design Rules Checking off.

## Creating a fanout pattern

**BGA button > Add Route button**

To begin copying traces, create a fanout pattern to serve as the template for the copies.

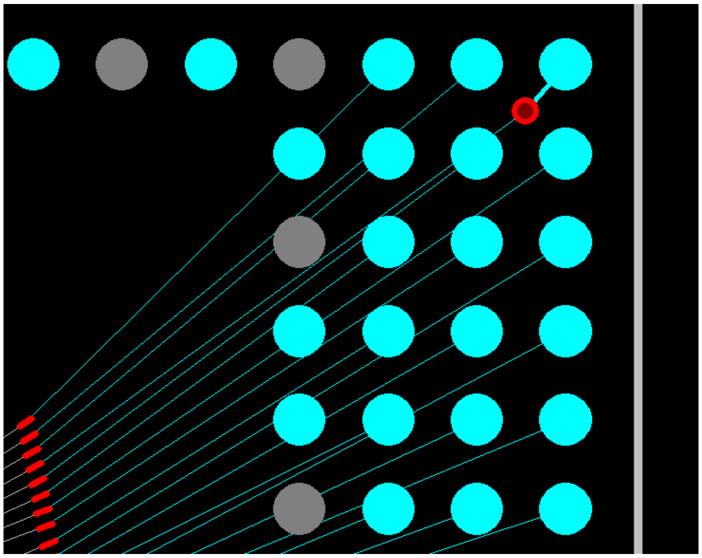
### To create the pattern:

1. On the standard toolbar, select the **BGA Side** from the Layer list.
2. Select the pin attached to the upper right-most ball pad with a connection.

   Result: A new trace starts and attaches to the pointer.

3. Move the pointer to the position shown in the graphic below. Ctrl+click to end the trace with a via attached.

### Via position:



### To copy traces:

1. Revert to Select mode by clicking the **Select** button on the BGA toolbar.
2. With nothing selected, right-click and click **Select Traces/Pins/Unroutes**.
3. Position the pointer over the diagonal trace segment and click to select it.
4. Position the pointer over the via and Ctrl+click to add it to the selected objects.
5. On the **Edit** menu click **Copy**. A copy of the trace attaches to the pointer.
6. Right-click, and then click **Next Base Point**.
7. With a trace attached to the pointer, click over the pin below the current BGA pad to add a copy of the trace. A copy is added and the pointer jumps to the next pad automatically.
8. Right-click and click **Repeat**..
9. On the Copy Route Repeat dialog box, type **10** and then click **OK**.

   Result: Ten copies of the trace and via are added.
10. Right-click, and click **Cancel**.

    Tip: You cannot use Esc to exit copy routines. You must wait until the routine completes. If the routine produces unexpected results, you can click the Undo button to undo any actions.
11. Do not save a copy of the design.

**You completed the connecting without a netlist tutorial.**

# Connecting With the Route Wizard

The BGA Route Wizard is an automatic connection-generating and autorouting tool. It eliminates repetitive and often tedious design tasks. The BGA Route Wizard has two modes of operation:

- **Generate Connections** Automatically generates connections between substrate bond pads and BGA package pads. It calculates the shortest connection lengths with the fewest number of crossovers.
- **Generate Connections and Route** Generates a netlist using the same features as Generate Connections mode, and routes the design automatically using established design rules.

### In this lesson:

- Starting the BGA Route Wizard
- Setting routing options
- Choosing pads to process
- Setting BGA fanout options
- Running the BGA Route Wizard

### Restriction

- This tutorial requires the Radial Placement, Auto-connect, BGA Fanout, General Editing and Plating Tails licensing options. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_6.pcb** in the \PADS Projects\Samples folder.

## Starting the BGA Route Wizard

**BGA button > Route Wizard button**

- In the Action area of the BGA Route Wizard dialog box, select **Generate Connections and Route**.

  Result: The Routing, Select Pads, and BGA Fanouts tabs are enabled.

## Setting routing options

### BGA Route Wizard > Routing tab

Use the Routing tab to set trace width, plating tail, and other routing-related options. Options for net name preferences are also available in this tab.

**To set routing options:**

1. In the Include area, make sure **Plating Tails** is selected.
2. In the Plating Tails area, in the Tail Length box, type **1** to replace the default value.
3. In the Net Name Preferences area select **Derive Net Name from Pin Function**.

## Choosing pads to process

### BGA Route Wizard > Select Pads tab

Use the Select Pads tab to specify which package pads and substrate bond pads to process when building connections and routing. You can click individual pads, pads connected to nets, or entire sides of the package. By default, processing includes all package and substrate bond pads.

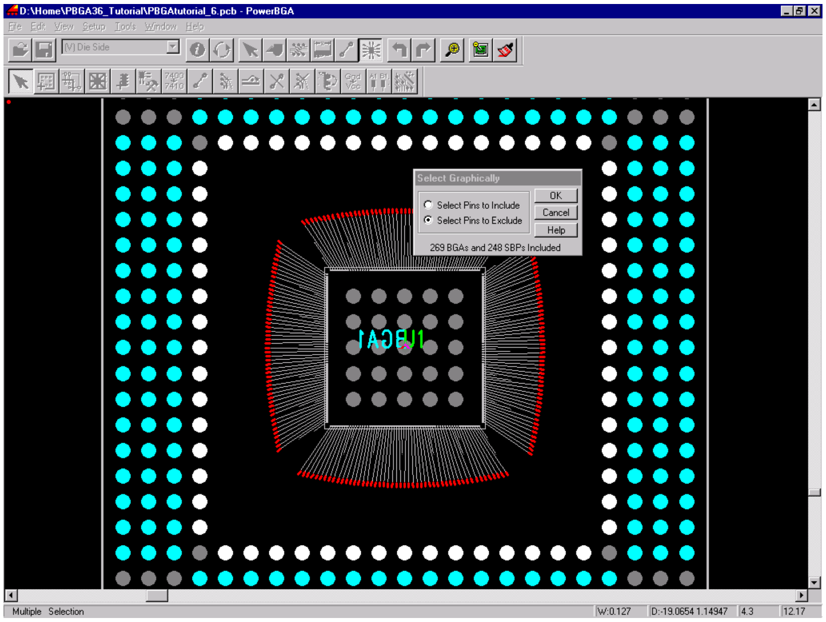
**To exclude the center BGA pads:**

1. Click **Select Graphically**.

   Result: The BGA Route Wizard dialog box closes and the Select Graphically dialog box appears.
2. Click **Select Pins to Exclude**.
3. Drag the pointer from **-10.7**, **-9.4** to **-9.5**, **9.6** to area-select the ball pads on the left side of the inside ring.

   Result: The ball pads highlight. See the white pads in the example below.

### Pads to select:




4. Repeat step 4 to select the remaining ball pads on the inside ring. Remember to press **Ctrl** before selecting because you are adding to the selection set.

5. Click **OK** in the Select Graphically dialog box. The Select Graphically dialog box closes, the BGA Route Wizard dialog box appears, and the pins are added to the Excluded list in the BGA Pins area.

### To exclude pads assigned to the GND and PWR nets:

1. In the Substrate Bond Pads area, on the Included list, click the **Net** column to sort the list by net.

   Result: The pads now appear first in the list.

2. Ctrl+click all substrate bond pads connected to **GND** and **PWR**.

3. In the Substrate Bond Pads area, click **Exclude**. This moves the pads attached to GND and PWR to the Excluded list in the Substrate Bond Pads area.

4. Repeat these steps for the BGA Pins you want to exclude.

5. Browse the Included list for both substrate bond pads and BGA pins to verify that no PWR or GND nets appear in the list.

## Setting BGA fanout options

**BGA Route Wizard > BGA Fanouts tab**

You use the BGA Fanouts tab to control the style, direction, and trace width of BGA Fanouts.

### To set these options:

- In the Fanout Style list, select **Diagonal**.

## Running the BGA Route Wizard

**BGA Route Wizard > Run button**

Create connections, traces and plating tails.

### To run the Generate Connections and Route function:

**Tip:** You can interrupt processing at any time by pressing the Esc key.

1. When processing completes, a report appears in your default text editor. Review the report then close it when you are done.
2. Do not save a copy of the design.

**You completed the connecting with the route wizard tutorial.**

# Adding Teardrops

PADS Layout includes features to generate teardrops automatically. In this lesson, you'll assign teardrop preferences and add teardrops to all trace end points on the design.

### In this lesson:

- Enabling teardrop generation
- Modifying teardrop geometry

### Restriction

- This tutorial requires the Analog licensing option. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_7.pcb** in the \PADS Projects\Samples folder.

## Enabling teardrop generation

**Tools menu > Options > Routing tab**

You enable teardrop generation in the Routing tab of the Preferences dialog box. Once you enable this option, a teardrop is added automatically to each trace end attached to a via or pad. Disabling the teardrop generation removes the teardrops previously generated.

### To enable teardrop generation:

- Select **Generate Teardrops**, and then click **OK**.

  Result: Teardrops are immediately added to each route end at vias and pads.

## Modifying teardrop geometry

Now that you've learned how to generate and remove teardrops, you can adjust the teardrop geometry. In this section, you'll learn how to change the geometry of one or all teardrops by changing the Query/Modify Teardrop settings.

### Resize the view

1. Press **Ctrl+B** to view the entire design.
2. Zoom into an area of the design, centering the view on a BGA pad and its fanout to the via.

### Modify the teardrop parameters

You can change teardrop parameters by selecting a trace segment with teardrops at one end of the segment and then open the Q/M Teardrop on Traces dialog box. Use this dialog box to make adjustments to the selected teardrops attached to the segment, apply changes to all teardrops, or apply them to teardrops on the current layer.

#### To modify teardrops:

1. With nothing selected, right-click, and then click **Select Traces/Pins**.
2. Select the diagonal segment attached to a BGA pad, right-click, and then click **Q/M Teardrop**. The diagonal segment may be difficult to see. It runs through the center of the teardrop.
3. On the Q/M Teardrop on Traces dialog box, click **All** in the Apply to area to apply the changes to all teardrops.
4. In the Shape area, click the **Line** button.
5. In the Length Ratio and Width Ratio boxes, type **100**.
6. In the Parameters area, select the **Auto Adjust** check box, until the checkmark appears in black instead of in gray.
7. Click **OK**.

   Result: Once the process is completed, all teardrops update to resemble a tapered trace.
8. Do not save a copy of the design.

**You completed the adding teardrops tutorial.**

# Creating Die Flags and Power Rings

PADS Layout provides functionality that enables parametric construction of die flags and power rings.

### In this lesson:

- Setting the copper hatch grid
- Defining the die flag
- Adding power rings

### Restriction

- This tutorial has no restrictions.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_8.pcb** in the \PADS Projects\Samples folder.

## Setting the copper hatch grid

**Tools menu > Options > Grids tab**

### To set the copper hatch grid:

1. In the Hatch Grid area, in the Copper and Keepout boxes, type **0**.
2. Click **OK**.

## Defining the die flag

**BGA button > Die Flag Wizard button**

Zoom into an area of the design, centering the view on the die and the wire bond fanout pattern.

1. With nothing selected, right-click and click **Select Component**.
2. Select component **U1**.
3. In the Die Flag and Rings area, ensure that **Die Flag** is selected.
4. In the Layer list, select **Die Side**.
5. In the Shape list, select **Rounded Rectangle**.

6. In the Spacing box, type **.25**.
7. In the Width box, type **.5**.
8. In the Net list, select **GND**.
9. In the Spokes area, do the following:
    - In the Number list select **8**.
    - In the Width box type **.6**.
    - In the Miter Size box type **.1**.
    - Select Project from corners.
10. In the Center Paddle area, in the Coverage box, type **50**.

Result: The workspace will update to show the die flag parameters.

## Adding power rings

1. In the Die Flag and Rings area, click **Add**.

   Result: A new ring named Ring1 is added.
2. In the Die Flag and Rings area, select **Ring1**.
3. In the Layer list, select **Die Side**.
4. In the Shape list, select **Rounded Rectangle**.
5. In the Spacing box, type **.25**.
6. In the Width box, type **.5**.
7. In the Net list, select **PWR**.
8. Click **Create** to generate the design copper for the die flag and power ring.

   Tip: To remove the die flag and power rings once they have been created in design copper set the selection filter to Select Shapes, select a window encompassing the die flag and power rings, and press Delete.
9. Do not save a copy of the design.

### You completed the creating die flags and power rings tutorial.

# Connecting Power and Ground Pads

### In this lesson:

- Highlighting the power net and pins
- Setting options
- Routing the power connections
- Creating via fanouts for ground ball pads

### Restriction

- This tutorial requires the General Editing licensing option. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_9.pcb** in the \PADS Projects\Samples folder.

## Highlighting the power net and pins

**View menu > Nets**

1. In the View List area, select the **PWR** net.
2. In the Color by Net area, select bright purple.
3. Click **OK**.

## Setting options

To ease the positioning of traces and vias, it is best to set a routing grid and a few other options.

Tip: Messages appear in the status bar as you complete these steps.

### To set the options:

1. Type **g0** and press **Enter** to set the working grid to 0.
2. Type **gv.05** and press **Enter** to set the via grid to .05.
3. Type **ad** and press **Enter** to set the angle mode to diagonal.

4. Type **e** and press **Enter** to set the end via mode. The message *End Via Mode Set* appears in the status bar. If another message appears, continue to type **e** and press **Enter** until the message *End Via Mode Set* appears.

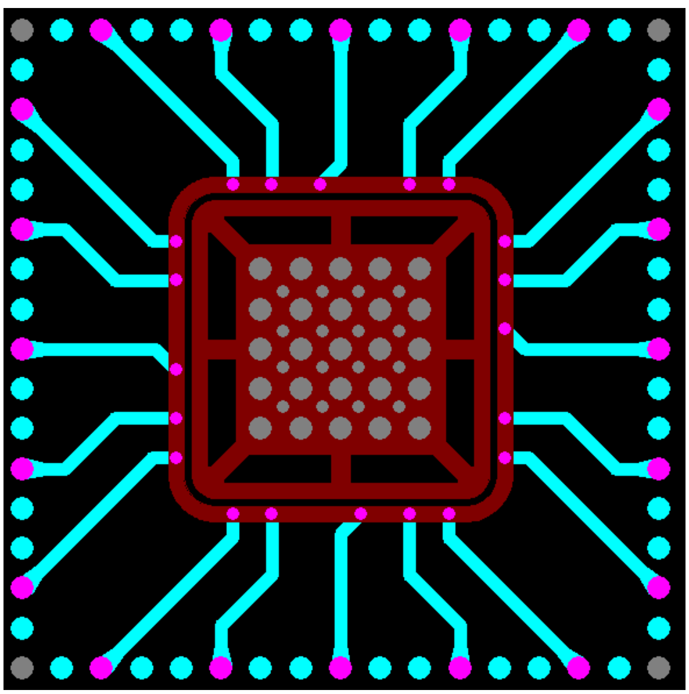
## Routing the power connections

**Setup menu > Display Colors**

1. Zoom into the lower left quadrant of the design.
2. Set the Die Side traces to **Black** to simplify the design.
3. Click **OK** to close the dialog box.

4. On the BGA toolbar, click the **Add Route** button.
5. Using the modeless Search command, type **s bga1.v4** and press **Enter**. Press **Spacebar** to start routing.
6. Right-click and click **Width**.
7. Type **.4** in the Modeless Command dialog box and press **Enter**.

Power connections:



8. Route the connection over to the power ring as shown.
9. Use Ctrl+click to end with a via going up to the power ring.
10. Repeat the routing step to connect all the **PWR** ball pads to the power ring. Because the design is symmetrical, you can use the copy command once you have made a few connections.

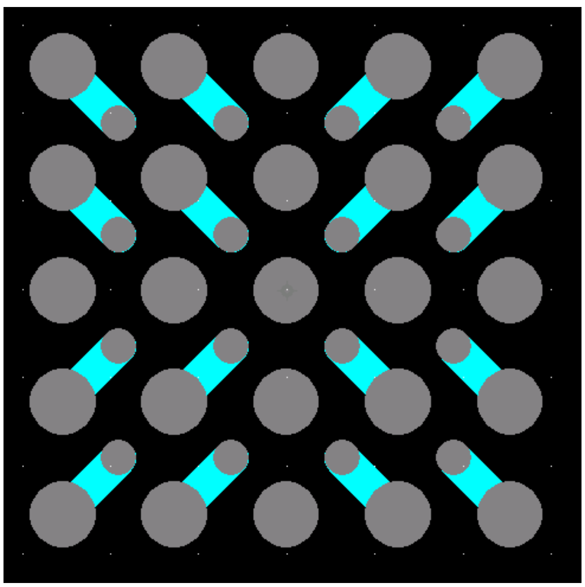
Tip: Remember to change the traces on Die Side back to red so you can see them.

## Creating via fanouts for ground ball pads

**Setup menu > Display Colors**

1. Below the Design Items area, clear the **Copper** check box and click **OK**.
2. Zoom into the center ball pad array.
3.  On the BGA toolbar, click the **Add Route** button.
4. Type the modeless Search command **s bga1.p10** and press **Spacebar** to start routing.
5. Right-click and click **Width**.
6. Type **.4** into the Modeless Command dialog box and press **Enter**.

GND connections:



7. Route the connection as shown above.
8. Ctrl+click to end with a via.
9. Repeat the routing step to fan out the **GND** as shown. Remember you can copy the objects, but ensure you set the selection filter to **Traces/Pins** first.
10. On the Setup menu click **Display Colors**.
11. Below the Design Items area, select the **Copper** check box and click **OK**.
12. Do not save a copy of the design.

**You completed the connecting power and ground pads tutorial.**

# Creating Copper Areas

### In this lesson:

- Updating design rules and preferences
- Adding the GND copper area

### Restriction

- This tutorial requires the Copper Flood and Drafting Editing licensing options. On the Help menu, click **Installed Options** to determine whether you can proceed.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_10.pcb** in the \PADS Projects\Samples folder.

## Updating design rules and preferences

**Setup menu > Design Rules > Rules dialog box > Default button > Default rules dialog box > Clearance button**

Before generating copper areas you must increase the rules.

### To set the rules:

1. In the Clearance Rules dialog box, in the Clearance area, set the rules Copper-Trace, Copper-Via, Copper-Pad and Copper-SMD to **.2**.
2. Click **OK**, and close the Default Rules and Rules dialog boxes.
3. On the Setup menu, click **Preferences**.
4. Click the **Thermals** tab.
5. In the Non-drilled Thermals area, select **Flood Over** to allow flooding over vias.
6. Click the **Drafting** tab.
7. In the Flood area, in the Min Hatch Area box, type **.6**.
8. Click **OK**.

## Adding the GND copper area

### To add the copper area:

1. Drafting button > Copper Pour button .
2. On the **View** menu, click **Extents** to view the entire design.
3. Verify the active layer is set to **BGA Side**.
4. Right-click and click **Rectangle**.
5. Type the modeless Search command **s bga1.ac1** and press **Enter**. Press **Spacebar**.
6. Type the modeless Search command **s bga1.a23** and press **Enter**. Press **Spacebar**.

## Modifying the copper pour shape

### To modify the copper area:

1. Select button .
2. With nothing selected, right-click and click **Select Shapes**.
3. Select the Copper Pour Shape created in the previous step.
4. Right-click and click **Query/Modify**.
5. In the Q/M Drafting dialog box, in the **Width** box, type **.15**.

## Assigning a net to the copper

### To assign the net:

1. In the **Net** area, select **GND**.
2. Click **Apply**.
3. Click the **Preferences** button.
4. Select the **Flood Over Vias** check box and click **OK**.
5. Click **Yes** in the Proceed with Flood message box.
6. Close the Q/M Drafting dialog box.
7. Do not save a copy of the design.

**You completed the creating copper areas tutorial.**

# Creating a Wire Bond Diagram

A typical requirement for documenting an advanced packaging design is to create a wire bond diagram. A wire bond diagram presents a picture of the die pads, wire bonds, and substrate bond pads. In addition, for each wire bond the name of the BGA pad to which it's connected appears adjacent to the substrate bond pad. In this tutorial, you will use the wire bond diagram command to add the BGA pad name labels to the design.

### In this lesson:

- Adjusting color settings
- Creating the wire bond diagram

### Restriction

- This tutorial requires the Drafting Editing and General Editing licensing options. On the Help menu, click **Installed Options** to determine whether you can proceed:

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_11.pcb** in the \PADS Projects\Samples folder.

## Adjusting color settings

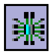
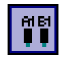
**Setup menu > Display Colors**

To demonstrate wire bond diagram, clear the display of those items not relevant to the wire bond diagram.

### To adjust color settings:

1. Clear the **Traces**, **Vias**, **Copper**, and **Errors** check boxes under those self-titled columns. Notice how the entire column of color tiles disappears.
2. Clear **BGA Side** next to that self-titled row. Notice how the entire row of color tiles disappears.
3. Select the **black** (same as the background color) color box in the Selected Color area, and then click **Connection** in the Other area.
4. Click **OK** to apply the color changes and close the Display Colors dialog box.

## Creating the wire bond diagram

**BGA button**  **> Wire Bond Diagram button** 

You use the wire bond diagram mode to define the parameters for the generation of the BGA pad name labels. You can apply this mode to a single pad or to all pads on a component.

### To start the command:

1. Press **Ctrl+B** to view the entire design.
2. Zoom into the top row of substrate bond pads.
3. With nothing selected, right-click and click **Select Components**.
4. Select one of the substrate bond pads.

   Result: The Add BGA Pin Labels dialog box appears.

## Setting the wire bond diagram preferences

The preferences in the Add BGA Pin Labels dialog box let you specify which pad label to add for those die pads connected to more than one BGA pad. You can also set size and layer assignment preferences for the text label to add.

### To assign the preferences and add the labels:

1. In the Width box type **.01** to assign a text width.
2. In the Height box type **.12** to assign a text height.
3. Click **OK**.

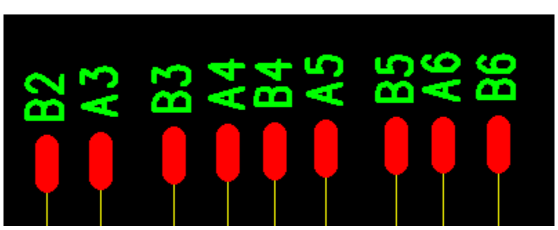
   Result: A label appears on each substrate bond pad.

## Rotating and moving the text labels

**Select button** 

After you add the labels, you must rotate and move them as shown in the following graphic.

### Label position:




1. With nothing selected, right-click and click **Select Documentation**.
2. Type **g0** and press **Enter** to set the working grid to 0.

3. Type **dro** and press **Enter** to turn off Design Rules Checking.
4. Area-select all the BGA labels on the top of the die.
5. Right-click, and then click **Rotate 90**.

   Result: All of the text labels are rotated 90 degrees counterclockwise.
6. With the text labels still selected, right-click, and then click **Move**. All of the labels now move as a group and are attached dynamically to the pointer.
7. Move the pointer and position the labels so they are adjacent to their respective substrate bond pads and click to place them.
8. Repeat these steps until you position all of the labels.
9. Do not save a copy of the design.

**You completed the creating a wire bond diagram tutorial.**

# Creating Wire Bond Reports

A supplied Basic script enables you to create two different report files in Microsoft Excel:

- A BGA pin (solder ball) to die function name report
- A die function name to BGA pin report

The output is sorted based on the report type you select. The reports perform connectivity reporting, verification, and die function name to net mapping.

**In this lesson:**

- Creating a wire bond report

**Tip:** The report appears in a text file automatically if you do not have Excel on your system.

### Restriction

- There are no restrictions for this tutorial.

### Preparation

If it is not already running, start PADS Layout and open the file named **PBGAtutorial_12.pcb** in the \PADS Projects\Samples folder.

## Creating a wire bond report

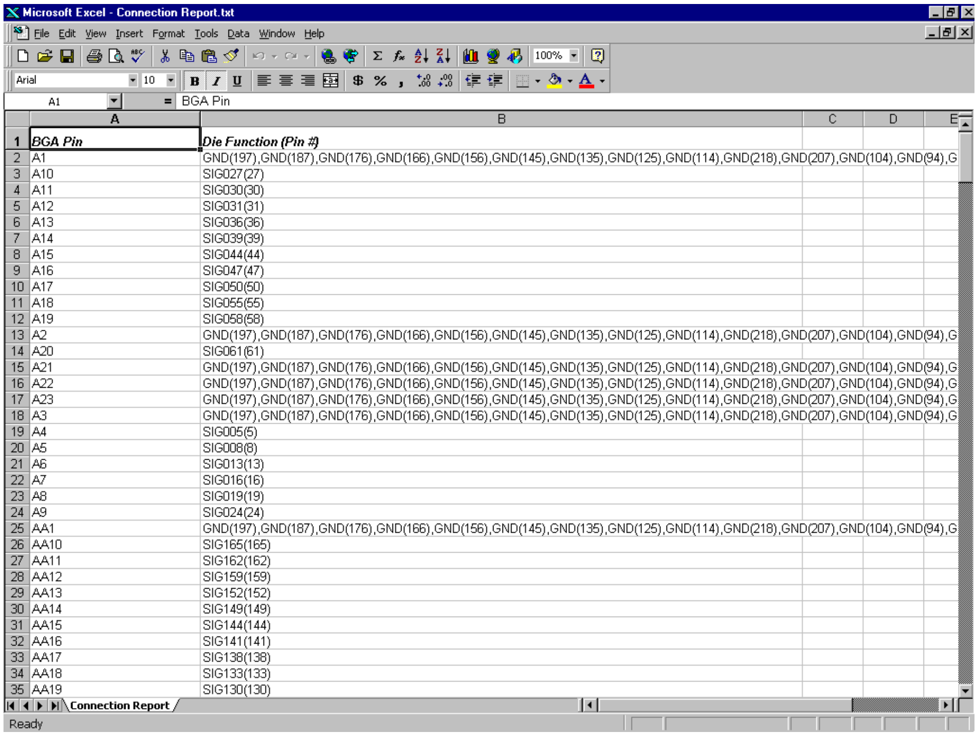
**Tools menu > Basic Scripts > Basic Scripts**

**To create a report file:**

1. In the Basic Scripts dialog box, select **BGA Wirebond Report** and click **Run**.

   **Result:** An Automation Utilities dialog box appears.
2. Select **BGA Pin to Die Function Name Report**.
3. Select **Include Die Pin Numbers**.
4. Click **OK** to start the report. The message *Depending on the complexity of your BGA design, this may take a moment or two* appears.
5. Click **OK** to continue. The connection report is generated and appears in Excel, as in the following graphic.

## The die function report:



| | BGA Pin | Die Function (Pin #) |
|---|---|---|
| 2 | A1 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 3 | A10 | SIG027(27) |
| 4 | A11 | SIG030(30) |
| 5 | A12 | SIG031(31) |
| 6 | A13 | SIG036(36) |
| 7 | A14 | SIG039(39) |
| 8 | A15 | SIG044(44) |
| 9 | A16 | SIG047(47) |
| 10 | A17 | SIG050(50) |
| 11 | A18 | SIG055(55) |
| 12 | A19 | SIG058(58) |
| 13 | A2 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 14 | A20 | SIG061(61) |
| 15 | A21 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 16 | A22 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 17 | A23 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 18 | A3 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 19 | A4 | SIG005(5) |
| 20 | A5 | SIG008(8) |
| 21 | A6 | SIG013(13) |
| 22 | A7 | SIG016(16) |
| 23 | A8 | SIG019(19) |
| 24 | A9 | SIG024(24) |
| 25 | AA1 | GND(197),GND(187),GND(176),GND(166),GND(156),GND(145),GND(135),GND(125),GND(114),GND(218),GND(207),GND(104),GND(94),G |
| 26 | AA10 | SIG165(165) |
| 27 | AA11 | SIG162(162) |
| 28 | AA12 | SIG159(159) |
| 29 | AA13 | SIG152(152) |
| 30 | AA14 | SIG149(149) |
| 31 | AA15 | SIG144(144) |
| 32 | AA16 | SIG141(141) |
| 33 | AA17 | SIG138(138) |
| 34 | AA18 | SIG133(133) |
| 35 | AA19 | SIG130(130) |

6. After reviewing the report, you can save it by clicking **Save As** on the **File** menu. Select a name and location for the file.
7. On the File menu, click **Exit** to return to PADS Layout.
8. Click **Close** to close the Basic Scripts dialog box.
9. Do not save a copy of the design.

## You completed the creating wire bond reports tutorial.