



## Board Shape & Sheet Basics

### Summary

Tutorial

TU0110 (v1.0) December 9, 2003

This tutorial covers board shapes, sheets, templates and keepouts used in DXP's PCB Editor.

The board shape defines the boundary, or extents, of the board in the PCB Editor. The board shape may also be referred to as a board outline and is essentially a closed polygon. It initially displays as a black area with the visible grid on by default when you create a new PCB document. It is used by DXP to determine the extents of the power planes for plane edge pull back, used when splitting power planes and for calculating the board edge when outputting design data to other tools, such as the 3D viewer.

When a new board file is created by selecting **File » New » PCB** from the menus, a default board shape is created, sized 6,000 x 4,000 mils. The board shape can be resized, or redefined, using the commands in the **Design » Board Shape** sub-menu. PCB documents created using the PCB templates or the PCB Board Wizard have the board shape already correctly sized.

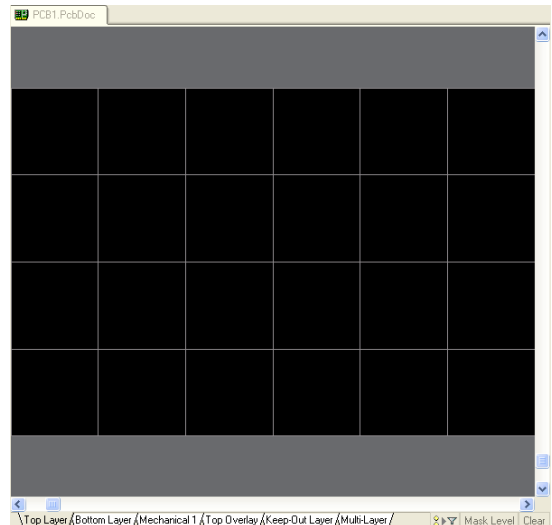
### Modifying a board shape

You can change a board shape by redefining (redrawing) it or by moving the vertices. You can move the board shape around the sheet as well, with or without any placed objects.

You can change the color of the board shape by selecting **Design » Board Layers & Colors** (shortcut key L) and selecting a new color for the Board Area in the System Colors section of the *Board Layers & Colors* dialog.

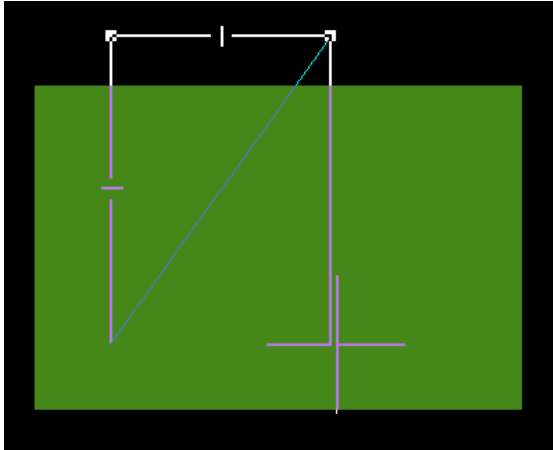
### Redefining a board shape

You can redefine the board shape if it does not already exist or you want to draw it again from scratch. If you need to change the entire board shape, complete the following steps.



## Board Shape & Sheet Basics

1. Select **Design » Board Shape » Redefine Board Shape**. The cursor will change to a large cross, the background will change to black and the original board shape will be displayed in green.
2. Click (or press **ENTER**) to create the corners of the new board shape. Press the **SPACEBAR** to change the corner style while you are defining the board shape. The Status bar at the bottom of the design window helps to locate the co-ordinates of the corners.



3. When you have defined the board shape, right-click or press **ESC** to finish. There is no need to fully close the polygon as DXP will automatically complete the shape by joining the first point to the last point placed.

The visible grid will be drawn to fill the area defined by the new board outline.

### Defining the board shape from selected objects

You can define an enclosed boundary, using lines and arcs, on a mechanical layer (or any layer) and use these objects to define the board shape.

To define a board shape from selected objects:

1. Create an enclosed boundary on a mechanical layer that will define the board shape you require. Use the placement commands such as **Place » Line** or **Place » Arc** to create your new board shape.
2. Select the new board shape boundary only.
3. Select **Design » Board Shape » Define from Selected Objects** and the board shape will be redisplayed to fit the selected boundary objects.

### Moving board vertices

When modifying a board shape, e.g. resizing it, moving the board vertices will save you from having to redefine the entire board shape.

1. Select **Design » Board Shape » Move Board Vertices**. The board outline displays with editing handles and the cursor changes to a large cross, ready to select and move vertices.

2. Click on the vertex that you want to move and drag it to its new location.
3. You can create new vertices by clicking on the small crosses that appear midway along the line segments and dragging the new vertex into position.
4. Right-click or press ESC to finish your board shape.

## Moving the board shape

Using the Move Board Shape command to reposition the board shape will move the board outline only. Any components and connections already placed will not be affected.

If you need to reposition the board shape in relation to the design sheet, make sure the sheet is visible by enabling the **Display Sheet** option in the *Board Options* dialog (**Design » Board Options**). See *Using PCB sheets* in this tutorial for more information about using sheets in DXP.

To move a board shape only:

1. Select **Design » Board Shape » Move Board Shape** and the outline will appear floating on the cursor.
2. Drag the board shape to its new location and click to place.

To move a board shape along with any components and connections already placed:

1. Select all (**Ctrl+A**) or select the objects you need to move, including the board shape.
2. Click directly on a selected object and the cursor changes to a large arrow. Drag the selection bounding box to the new location on the sheet.

Alternatively, select the objects required and select **Edit » Move » Move Selection**. Click within the selection to define a reference point, move the selection and click to place.

## Using PCB sheets

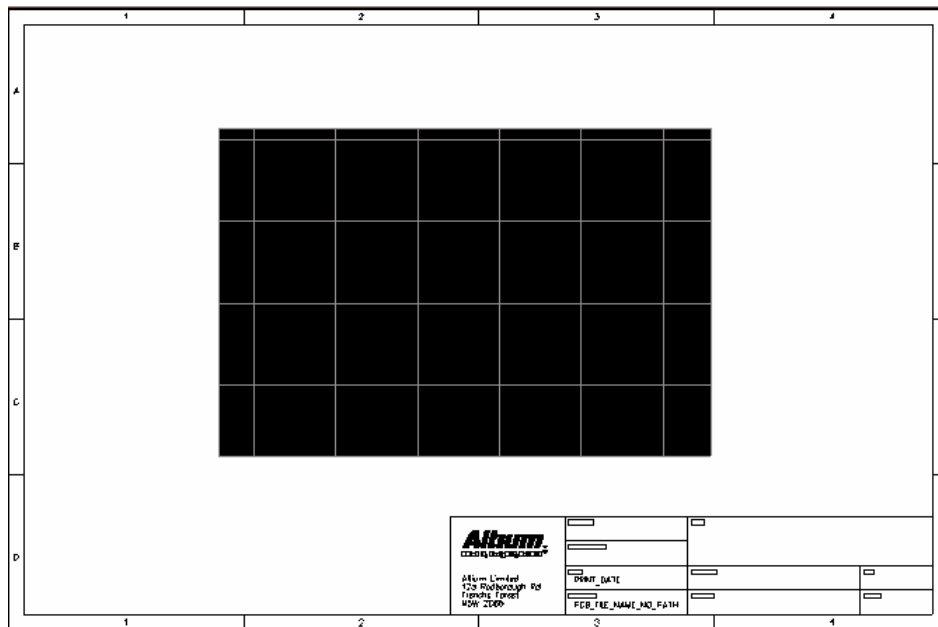
---

Sheets in the PCB Editor are a special drawing feature, controlled using the options in the *Board Options* dialog. When you create a new PCB file, a default sheet is automatically created with the default size of 10000 x 8000 mil. It is not shown initially but, when displayed, it appears as the white shape behind the board outline.

Most of the PCB example files supplied with DXP (`C:\Program Files\Altium2004\Examples`) display the board on a white sheet which includes a border, grid reference and title block that have been drawn on one of the mechanical layers, Mechanical16.

By placing objects on mechanical layers and then linking those layers to the sheet, you can create your own drawing templates which can be displayed or hidden. Sheets that include a border, grid reference and title block can be added to PCB files by copying from the existing PCB templates (`C:\Program Files\Altium2004\Templates`).

## Board Shape & Sheet Basics



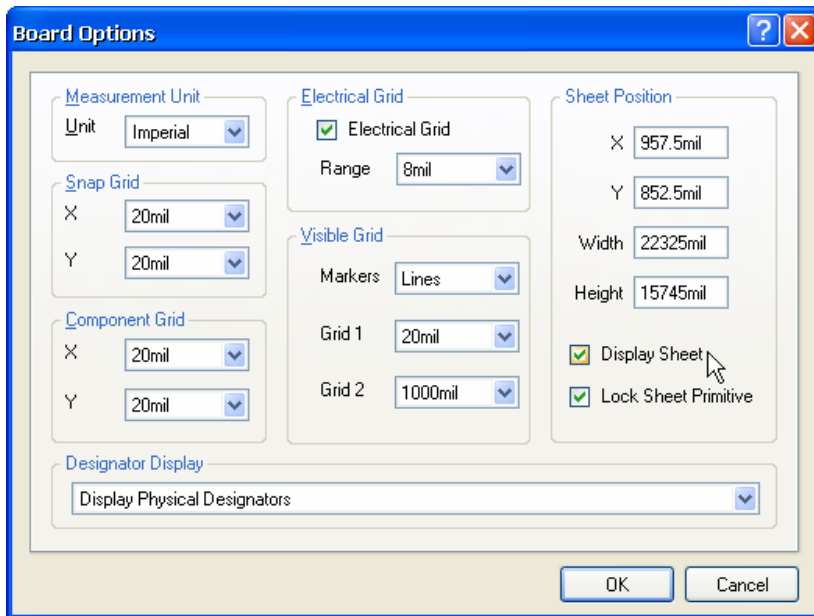
The sheet size and location of the sheet can be defined manually, or the sheet can be resized automatically to fit the objects on the linked mechanical layer when you select **View » Fit Sheet**.

### Displaying the sheet

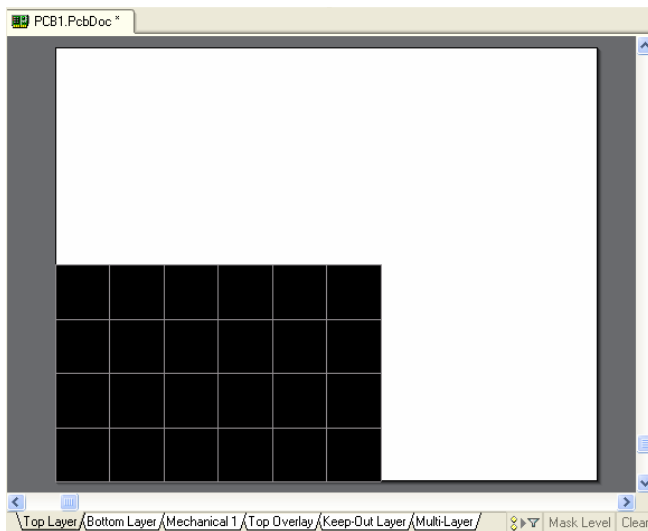
To make the sheet is visible in the PCB Editor:

1. Select **Design » Board Options** and select the **Display Sheet** option in the Sheet Position section of the *Board Options* dialog. Click **OK**.

The sheet can be hidden at any time by disabling the **Display Sheet** option. All linked mechanical layers will also be hidden.



2. Select **View » Fit Sheet** to display the sheet. Alternatively, use the shortcut keys, **V, H** (View Sheet) or **Z, S** (Zoom Sheet). A white space appears around the board shape with the default size of 10000 x 8000 mil.

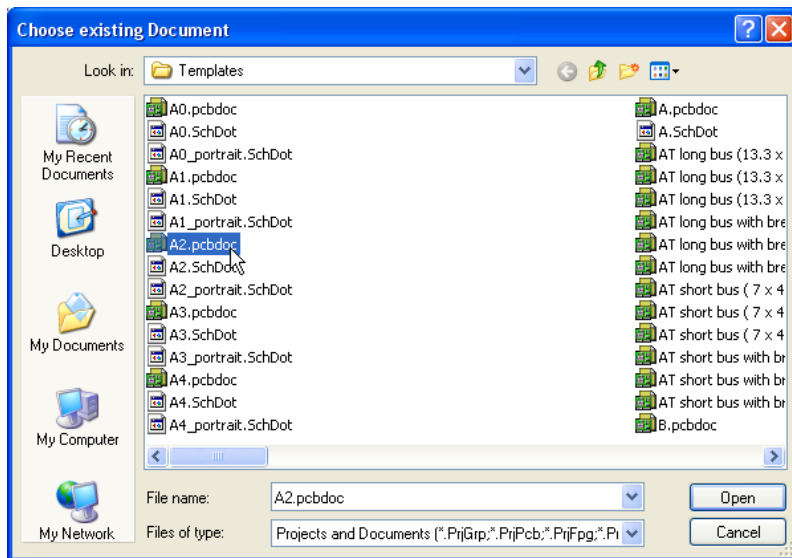


3. You can change the color of the sheet by selecting **Design » Board Layers & Colors** (shortcut key **L**) and selecting a new color for the Sheet Area and Sheet Line in the System Colors section of the *Board Layers and Colors* dialog.

### Adding a new sheet from a PCB template

You can add a new sheet, including the sheet border, reference grid and title block, at any time by copying the objects from the supplied DXP PCB template documents and pasting them into your PCB design document. DXP includes a set of pre-defined PCB templates located at `C:\Program Files\Altium2004\Templates`. Use the sheet size templates only, i.e. `A.pcbdoc` through to `A0.pcbdoc`.

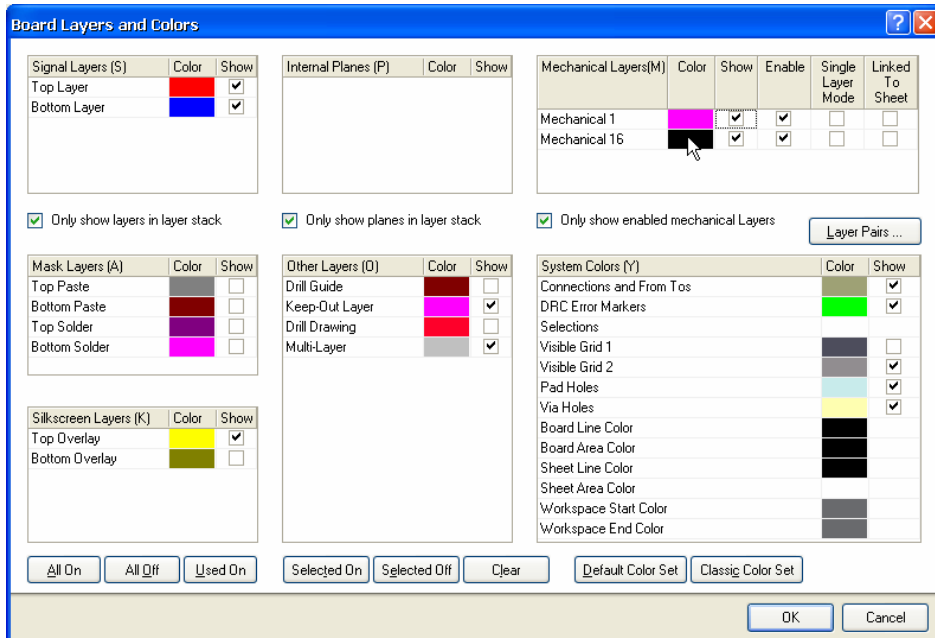
1. Open the PCB document that you want to add the new sheet size to. Make sure that the existing default sheet is displayed to help you place the new sheet by pressing **V, H** (view sheet) or **Z, S** (zoom sheet).
2. Open an existing PCB sheet template that will fit all the objects on your PCB, e.g. `A2.pcbdoc`. To do this, click on **PCB Templates** in the **New from Template** section of the **Files** panel. If this option is not visible, click on the up arrows to the right of each section in the Files panel to contract the other options. The *Choose existing Document* dialog displays.



Navigate to the PCB templates folder (`C:\Program Files\Altium2004\Templates`) and select `A2.pcbdoc` (for example) and click **Open**. The template is opened as a new PCB design document in the design window, named `PCB1.PcbDoc`.

3. Select all the contents of the template file (**Ctrl+A**) and copy (**Ctrl+C**) the contents to the clipboard. Click once to set a copy reference point. Close `Pcb1.pcbdoc` without saving.
4. Switch to your PCB document by clicking on its tab at the top of the design window. Paste the new sheet into the existing PCB using **Ctrl+V**. The contents of the template are pasted onto Mechanical16 layer.
5. Now we need to show the Mechanical16 layer and link it to the sheet. Select **Design » Board Layers & Colors** to display the *Board Layers and Colors* dialog. Click on **Show, Enable and Linked to Sheet**.

6. Also turn on **Single Layer Mode** to always show the sheet regardless of the status of Single Layer Mode as enabled in the **Display** tab of the *Preferences* dialog (**Tools » Preferences**). Click **OK** to close the dialog.



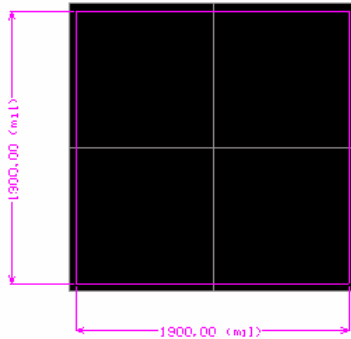
7. Finally we can size the sheet to include the sheet border. Press **V, H** (to view sheet) or **Z, S** (to zoom sheet). The sheet fits to the extents of the objects on the layer linked to the sheet, i.e. it fits to include the sheet border defined on the Mechanical16 layer.
8. You can now modify the title block, for example, by switching to Mechanical16 layer and adding or deleting objects. The sheet will resize to include all objects when you press **V, H** (view sheet) or **Z, S** (zoom sheet) again.

## Keepouts

The keepout is a separate entity to the board shape. A keepout shape, usually made up of placed keepout tracks, is required on the PCB Keepout layer if you intend to run the autorouter or autoplacement components. A keepout track is simply a track placed on the Keepout layer with the Keepout attribute enabled.

A keepout is automatically generated if you create the PCB by using the PCB Board Wizard where you can nominate the distance of the keepout from the edge of the board shape. The dimensions generated by the PCB Board Wizard measure the keepout not the board shape size. The diagram below shows an example of a keepout outline generated on the Keepout layer by the PCB Board Wizard, indented 100mil from the edge of the board shape of 2000 x 2000 mil.

## Board Shape & Sheet Basics



A board keepout is also automatically included when using a PCB manufacturers' template to create a new PCB, e.g. AT or Eurocard. The sheet size templates, e.g. A2.pcbdoc, do not include keepouts.

### Creating a keepout

Since a keepout is required if you wish to autoplace components or autoroute the board, you will need to add the keepout yourself if you create a new PCB in the following ways:

- selecting **File » New » PCB** from the menus, or clicking on **PCB File** in the **New** section of the **Files** panel,
- using the PCB sheet size templates, e.g. A2.pcbdoc, by choosing **PCB Templates** from the **New from Template** section of the **Files** panel.

Only a default board shape is created by these commands, so the keepout has to be added once the board shape has been defined.

To create a keepout by placing keepout tracks:

1. Click on the **Keep-Out** layer tab so you will be placing the keepout tracks on this layer only.
2. Select **Place » Keepout » Track**. Click to define the vertex points of the keepout and create a closed polygon.
3. When you have finished placing the keepout tracks, right-click or press **ESC** to exit track placement mode.

Now you have created your board shape, sheet and keepout, set the grids, layers and design rules and you are ready to start designing your board.

Track placement modes are available when placing a keepout.

Press **SHIFT+SPACEBAR** to cycle through the modes.

Press **SPACEBAR** to toggle between Start and End modes.

Use **BACKSPACE** to remove the last placed track segment.

Press **TAB** to display the Line Constraints dialog and change properties.



## Revision History

---

Date	Version No.	Revision
9-Dec-2003	1.0	New product release

Software, documentation and related materials:

Copyright © 2003 Altium Limited.

All rights reserved. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, DXP, Design Explorer, nVisage, Nexar, Protel, P-CAD, Tasking, CAMtastic, Situs and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.