# Allegro® X
# Design Publisher User Guide

**Product Version 23.1**
**September 2023**

cādence®

# Contents

**1**

# Introduction

The Allegro Design Publisher solution (referred to as the *Publish PDF* utility in this user guide) facilitates viewing of complex logical and physical designs made in Design Entry HDL or Allegro PCB Editor or Allegro Package Designer in a Portable Document Format (PDF) file.

The design is made viewable independent of the application. This feature is especially useful when you need to share complex schematic designs and physical design with experts who might not have design tools installed on their computers.

> *Important*
>
> A PDF document generated using the Publish PDF utility is only compatible with Adobe Reader 7.0 or later. You can download the free Adobe Reader from the Adobe website.

This guide covers the following topics:

- For information on exporting Design Entry HDL schematics see, Chapter 2, "Exporting Schematics to PDF"

- For information on exporting Allegro PCB Editor designs or Allegro Package Designs see, Chapter 3, "Exporting Physical or Packaged Designs to PDF."

- For information on using the generated PDF file see,Chapter 4, "Navigating through the PDF."

> *Important*
>
> The *Publish PDF* utility is available on the Windows and Linux platforms. You need to have the relevant license installed to use this utility.

## Overview

The Publish PDF utility generates outputs which can be viewed on all platforms that support display of PDF documents. Using this utility, a PDF document is generated without the use of

Acrobat Distiller, so you do not even require a license of Acrobat Distiller. Some of the other advantages of this utility are:

■    Design Entry HDL or Allegro PCB Editor expertise is not required for viewing schematic designs.

■    The design can be viewed independent of the platform as PDF documents are viewable on almost all platforms.

■    PDF documents are also easily transferable on Internet.

■    Password protection of the PDF file for enhanced security.

In a published PDF document of a logical or physical design, you can do the following:

■    Descend into hierarchical blocks of a schematic design

■    View the design using a Hierarchical Model-Tree navigator

■    Control the display of _layers_ and objects

■    View properties on components and nets just as you view them in the Attributes dialog box of Design Entry HDL

■    View properties on components, nets, pins, vias etc just as you view them Show Properties form in PCB Editor.

**Note:** The Allegro Publish PDF utility generates non-editable PDF documents meant for viewing only.

**2**

# Exporting Schematics to PDF

The Allegro Design Publisher solution (referred to as the *Publish PDF* utility in this user guide) facilitates viewing of complex designs made in Design Entry HDL schematic editor in a Portable Document Format (PDF) file. The design is made viewable independent of the Design Entry HDL application. This feature is especially useful when you need to share complex schematic designs with experts who might not have Design Entry HDL installed on their systems.

This chapter covers the following topics:

- Setting up Preferences for Publishing on page 10

- Exporting Layers to PDF on page 12

- Setting up Advanced PDF Options on page 16

- Publishing a Schematic Design as a PDF on page 22

- Publishing a PDF Document from the Console on page 27

- Publishing a PDF Document from the Command Line on page 28

- Specifying Options for Page Setup from the Command line on page 30

 *Important*

> The *Publish PDF* utility is available on the Windows and Linux platforms. You need to have the relevant license installed to benefit from this utility.

## ▐▌ Video

Watch this multimedia demonstration, *Publishing a Design as a PDF Document*.

## Descending into Hierarchical Blocks of a Design

You can descend into the hierarchical blocks of a design in the published PDF document just as you would do in a schematic. You can quickly navigate the schematic design using the *Descend* command that appears in the pop-up menu for a hierarchical block.

Descending into a hierarchical block in a schematic

Descending into a hierarchical block in the published PDF document
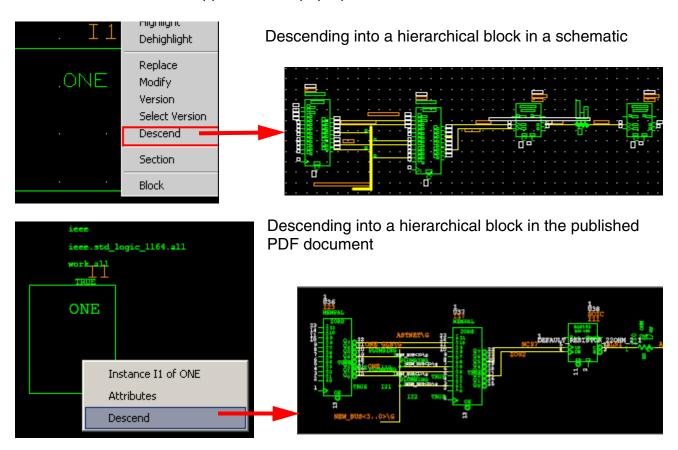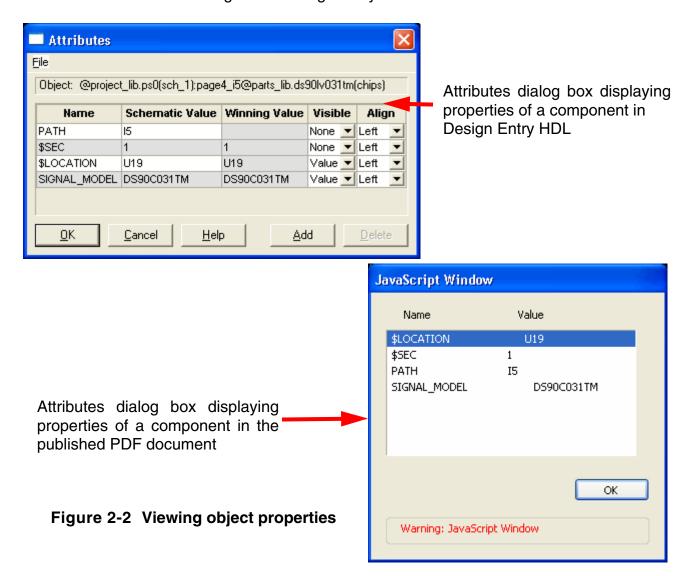
**Figure 2-1  Descending into a hierarchical design**

## Viewing Object Properties
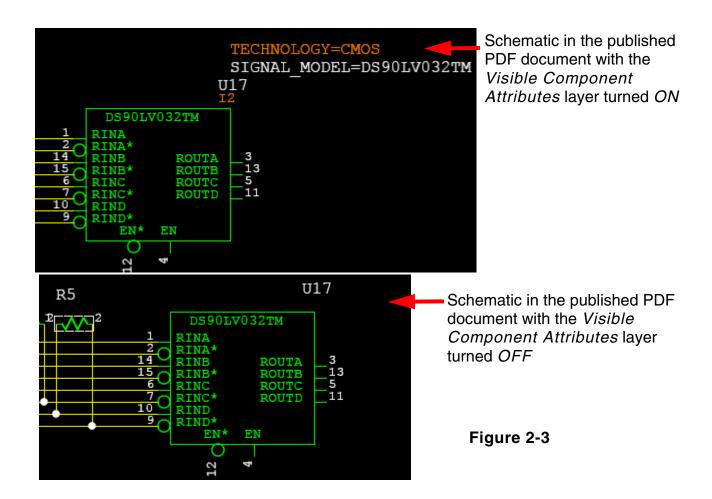
Properties of a schematic object appear in a dialog box in a published PDF document that is similar to the Attributes dialog box of Design Entry HDL.



Attributes dialog box displaying properties of a component in Design Entry HDL

Attributes dialog box displaying properties of a component in the published PDF document

**Figure 2-2  Viewing object properties**

If you want to display all the properties on the schematic canvas regardless of their Visibility definition in the Attributes dialog box of Design Entry HDL, select the *Export to File* and the

*Visibility* check boxes for the *Invisible Attributes* option in the *PDF Option* page of the Design Entry HDL Options dialog box.



Schematic in the published PDF document with the *Visible Component Attributes* layer turned *ON*

Schematic in the published PDF document with the *Visible Component Attributes* layer turned *OFF*

**Figure 2-3**

# Setting up Preferences for Publishing

Before you publish a PDF document of a schematic, you need to set the preferences to specify what information is to be exported to the PDF document. You can also control the

default visible information at the time of document loading by setting the publishing preferences.
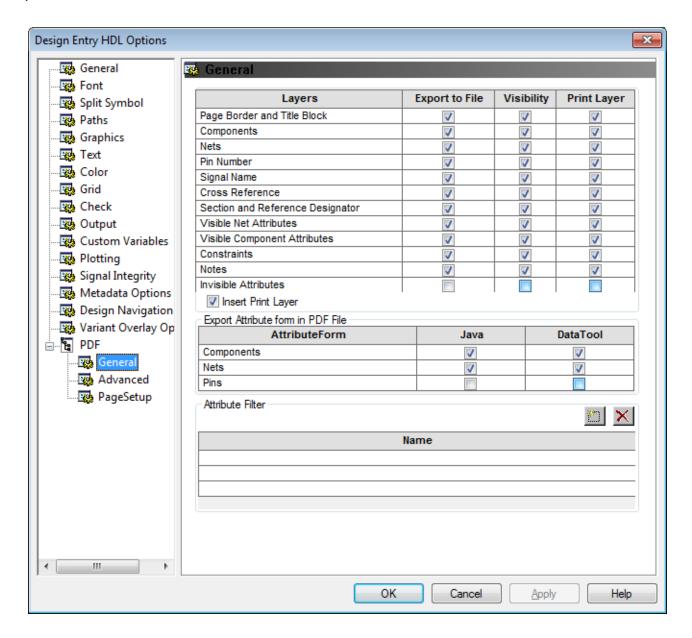


**Figure 2-3  Design Entry HDL Options - PDF Option Dialog Box**

**Note:** You can also access this dialog by clicking the *Setup* button in the Publish PDF Dialog Box which is launched after you run the command to publish the PDF using the *File — Publish PDF* menu command.

# Exporting Layers to PDF

This dialog box displays the layers and the options for exporting specific layers and specific properties to the published PDF document.

| Option | Description |
|---|---|
| Layers | Specifies the layer to be exported and made visible in the published PDF document. |
| Export to File | Allows you to specify the information to be exported to the published PDF document. This option can also be set by using the EXPORT directive in the START_PDF section of the project's .cpm file. The value of this directive identifies which layers are exported to the published PDF document and which are not. |
| Visibility | Displays or hides the selected layer. This option can also be set by using the VISIBLE directive in the START_PDF section of the project's .cpm file. The value of this directive identifies which layers are visible in the published PDF document and which are not. |
| Print Layer | Incorporates the printing feature in the viewable PDF. This option solves the problem of keeping two versions of a PDF document, one for viewing and one for printing. This option incorporates the printing feature into the viewable PDF. This way you can take printouts of published PDF documents exactly as they appear on the viewable PDF document.<br><br>**Note:** A viewable PDF depicts the schematic view exactly as it appears in Design Entry HDL, including the appropriate color coding for components, wires, signals, and page borders. |
| Insert Print Layer | Works in conjunction with the Print Layer option. You select the Insert Print Layer check box to include a printable PDF version in the generated PDF. After that make the required selections in the Print Layer column. This defines the layers that will be available in the printed version of the PDF that you generate for the schematic. |

| Option | Description |
|---|---|
| Export Attribute form in PDF File | Lets you specify whether you want the Attribute dialog box to appear in the published PDF document for various schematic objects. This option can also be set by using the `ATTRIBUTEFORM` directive in the `START_PDF` section of the project's `.cpm` file. You need to add <u>decimal values</u> corresponding to Component, Net, and Pin to set the `ATTRIBUTEFORM` directive.

■ Component Attribute Form: Select this option to specify if you want the Attribute dialog box to be displayed for components in the published PDF document.

■ Net Attribute Form: Select this option to specify if you want the Attribute dialog box to be displayed for nets in the published PDF document.

■ Pin Attribute Form: Select this option to specify if you want the Attribute dialog box to be displayed for pins in the published PDF document.

The following two options are used to view the properties of nets, components, and pins:

■ Java: The Java object attribute dialog box is similar to that is currently used in Design Entry HDL. However, to view attributes of a number of objects, you need to select the objects one by one, right-click, and select *View* attributes.

■ DataTool: DataTool gives you an easier way of viewing object properties. This option is also enables you to find objects on the published PDF document and in the Model tree.

**Note:** if you do not export Net and Pin Attribute dialogs, no hyperlinks are created in the published PDF document. This feature optimizes the PDF file size. In case of Components, however, hyperlinks are generated showing the instance name and the *Descend* option in case of blocks, in the pop-up menu. |

| Option | Description |
|---|---|
| Attribute Filter | **Note:** Lets you specify property names which you do not want to export to the published PDF document. This way you can filter out undesirable properties from being exported to the published PDF document. Filtered attributes are not visible in the Attribute dialog box of a part, net, or pin in a published PDF document. The properties you specify here are written to the START_PDF section of the project's .cpm file. For example, if you decide to exclude the SIGNAL_MODEL property from the published PDF document, the following line is added to the .cpm file: |

```
'SIGNAL_MODEL'
```

You set the *Visibility* option to specify the default visibility of a layer in the published PDF document. If this preference is set, the corresponding layer will be visible in the document at the time of loading. For a layer to display, you need to select the corresponding *Export to File* option for the layer in addition to the *Visibility* option. You can also set the EXPORT and VISIBLE directives in the .cpm file to export and display specific layers. For each layer, a bit is set in the directive value. To set the bit, you need to add the decimal corresponding values of the layer for the EXPORT and VISIBLE directives. For example, if you want to export the Component (value = 2), Nets (value = 4), Pin Numbers (value = 8), and Signal Names (value = 16) layers, you would add the corresponding values for the layers and assign the resultant value, 30, to the EXPORT directive. The decimal value for each layer is listed in the table below.

| Layer | Description | Decimal Value of the Layer |
|---|---|---|
| Page Border & Title Block | Published to the PDF document, by default. | 1 |
| Component | Published to the PDF document, by default. | 2 |
| Nets | Published to the PDF document, by default. | 4 |

| Layer | Description | Decimal Value of the Layer |
|---|---|---|
| Pin Numbers | This attribute is attached to a pin as specified in the `chips.prt` file. Pin numbers are published to the PDF document, by default. Pin Numbers with '?' and '#' in their names are not exported to the published PDF document. | 8 |
| Signal Names | This attribute is attached to signals. Signal names are published to the PDF document, by default. | 16 |
| Cross Reference | This attribute lets you traverse cross-references within the schematic. Cross-references are translated to links on the published PDF document. By default, cross reference attributes are exported to the PDF document. | 32 |
| Section and Reference Designator attributes | Published to the PDF document, by default. | 64 |
| Visible Net Attributes | All the net attributes for which the visibility has been set to true in the Attributes dialog box. Published to the PDF document, by default. | 128 |
| Visible Component Attributes | All the component attributes for which the visibility has been set to true in the Attributes dialog box. Published to the PDF document, by default. | 256 |
| Constraints | By default, constraints are not published to the PDF document. Constraints which do not have placeholder in schematic are not exported to the PDF document. | 1024 |
| Notes | By default, notes are not published to the PDF document. | 2048 |

| Layer | Description | Decimal Value of the Layer |
|---|---|---|
| Invisible attributes | This option refers to the invisible properties of nets, components, and pins. This layer corresponds to only those properties of nets, pins, and components which are not visible on schematic canvas. This layer appears in the published PDF document only if you export invisible properties to the PDF document. By default, invisible properties are not published to the PDF document. | 512 |

# Setting up Advanced PDF Options

Before publishing a PDF document you can also specify options, such as the PDF viewer in which you want to view the PDF document and whether the document is to be published in color or in black and white. All these tasks can be performed using the PDF—Advanced UI of the Design Entry HDL dialog box.

You can access the PDF Advanced option in one of the following ways:

❑ Choose *Tools – Options* and click *PDF — Advanced* in the Design Entry HDL dialog box.

❑ In the Publish PDF dialog box, click *Setup* and then click *PDF — Advanced*.
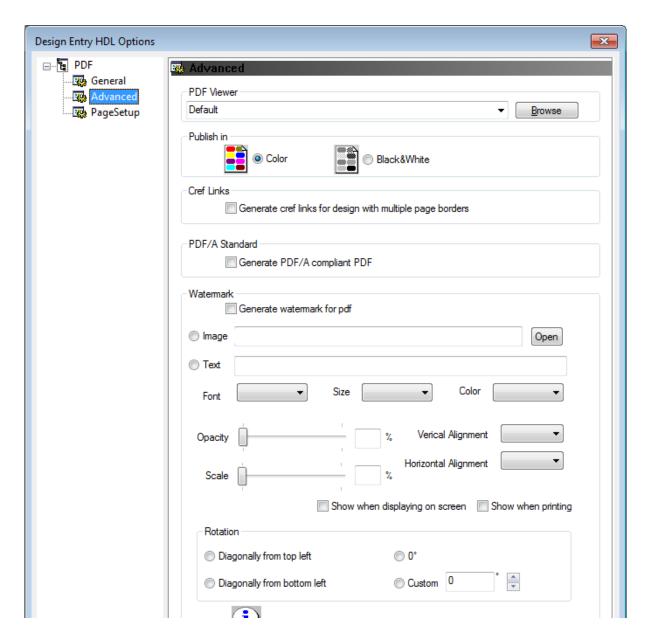


**Figure 2-4 Design Entry HDL Options - PDF - Advanced Dialog Box**

You can configure the following options in this page:

| Option | Description |
| --- | --- |
| *PDF Viewer* | Here you specify the path to the PDF viewer in which you want to view the PDF document. After the PDF generation is complete, you are prompted to view the generated PDF. By default, the PDF will open in Adobe Reader. If you want to use any other viewer, you can browse to the viewer's executable in this field. |
| *Publish In: Color \| Black&White* | This option allows you to generate a viewable PDF in either color or black and white. If you choose *Black&White*, on printing, the PDF will print in black and white. |
| Generate cref links for a design with multiple page borders | With this option, you can generate cross-reference links for designs which have more than one type of page border used in the design.<br><br>For information on navigating through cross-reference links in a PDF, refer to <u>Cross-References as Links</u> on page 66. |
| *PDF/A Standard: Generate PDF/A compliant PDF* | Choose this option if you want to print ISO 9005-1-compliant PDF files.<br><br>If you use the Cadence default font for your schematic design, that is, the Concept Font, DE-HDL prints the PDF/A compliant PDFs using the Courier New font. The font is embedded in the document.<br><br>If you use another font for your schematic design, PDF/A compliant PDFs are printed in that font, which is also embedded in the document. |
| *Watermark: Generate watermark for pdf* | Select this option if you want to apply a watermark (text or image) to your PDF file. For example, you could apply a "Confidential" watermark to pages with sensitive information. |

| Option | Description |
|---|---|
| *Watermark* | *Image*: Select the image that you want as a watermark in your PDF file. The image can be on your local drive, or a network drive, provided you have the required permission. |
| | **Note:** Only images with .jpg/.jpeg formats are supported. |
| | *Text*: Specify the text that you want as a watermark in your PDF file. |
| | *Font*: Choose the font for your watermark text. |
| | *Size*: Choose the font size for your watermark text. |
| | *Color*: Choose the font color for your watermark text. |
| | *Opacity*: Specify the degree of opacity for the watermark. On a scale of 0% to 100%, 0 would make the watermark fully transparent and 100 would mean that the watermark is fully opaque. |
| | *Scale*: Specify the scale factor for the watermark. |
| | *Vertical Alignment*: Specify the vertical placement of the watermark with respect to the document. The options are Top, Center and Bottom. |
| | *Horizontal Alignment*: Specify the horizontal placement of the watermark with respect to the document. The options are Top, Center and Bottom. |
| | *Show when displaying on screen*: Specify whether the watermark should be displayed when viewing the PDF on screen. |
| | *Show when printing*: Specify whether the watermark should be displayed in the printed PDF. |
| *Rotation:* | *Diagonally from top left:* Select this option to display the watermark from the top left. |
| | *Diagonally from bottom left:* Select this option to display the watermark from the bottom left. |
| | *0°*: Set the rotation of the watermark to 0° degrees. |
| | *Custom:* Specify a custom angle of rotation for the watermark (0 to 360 degrees). |

# Specifying Page Setup Options

In addition to the basic and advanced publishing options, publishpdf also supports parameters to specify page setup options, such as measurement unit, page size, orientation, margins, and scaling factor. At the time of generating a PDF document, you can specify the page size, margin, and orientation, and also scale the output size of the PDF document to be generated.

You can specify these options using one of the following methods:

■ Using the Page Setup UI

■ Specifying Options for Page Setup from the Command line

## Using the Page Setup UI

You can access the Page Setup option in one of the following ways:

❑ Choose *Tools – Options* and click *PDF—PageSetup* in the Design Entry HDL dialog box.

❑ In the Publish PDF dialog box, click *Setup* and then click *PDF — PageSetup*.



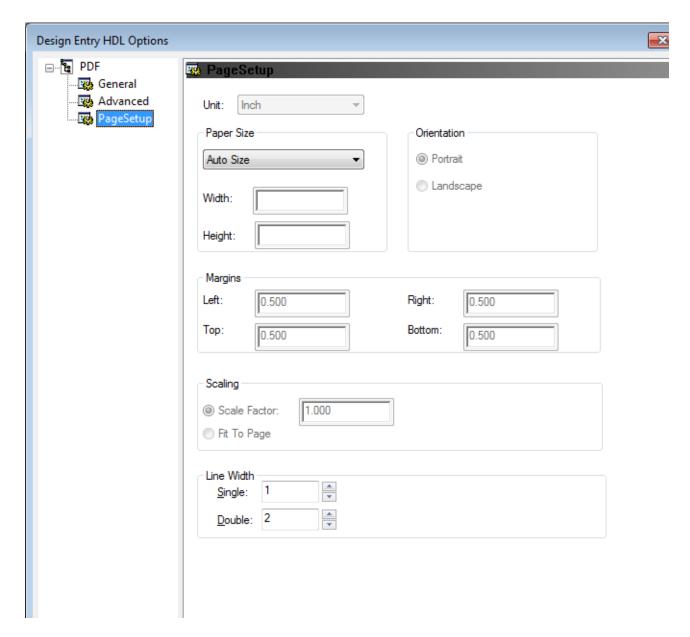**Figure 2-5  Design Entry HDL Options - PDF - Page Setup Dialog Box**

You can configure the following options in this dialog:

| Option | Description |
| --- | --- |
| Unit | Specify the measurement unit as either inches or millimeter. |
| Paper Size | Specify the paper size as one of the standard sizes or a custom size, wherein you also need to specify the paper Width and Height.<br><br>■ ANSI (A to E)<br><br>■ A (0 to 10)<br><br>■ B (0 to 10)<br><br>■ C (0 to 10)<br><br>■ Custom Size |
| Orientation | Specify that the page orientation, whether landscape or portrait. By default, the orientation is portrait. |
| Margins | Specify the left, right, top, and bottom margins of the page. |
| Scaling | ■ **Scale Factor:** Specify the scaling factor, in percentage, by which the page is to be scaled, reduced or enlarged.<br><br>■ **Fit to Page:** Specify that the document is to be scaled to fit on the page. |
| Line Width | ■ **Single:** Specify the line width to be used for thin wires while generating the PDF output file.<br><br>■ **Double:** Specify the width to be used for thicker wires while generating the PDF output file. |

# Publishing a Schematic Design as a PDF

After setting up the required preferences for exporting information, you can publish the schematic design as a PDF document.

■    Winning values are displayed in the attributes form. The same properties are displayed in the published PDF.

To publish a PDF document of a schematic, perform the following steps:

1. Choose *File — Publish PDF*.

   The Publish PDF dialog box is displayed.

   You can also publish a PDF of the schematic using a *console* command.
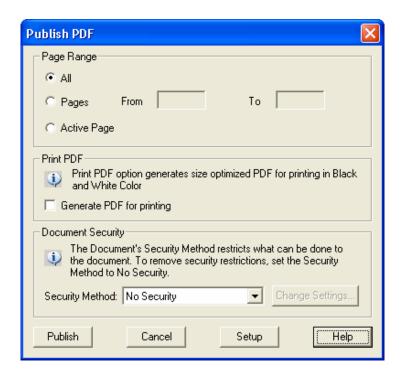


**Figure 2-6  Publish PDF Dialog Box**

2. Specify the pages or range of pages to publish in the *Page Range* section.

   You can choose to print the entire schematic, a specified range of schematic pages, or just the currently active page.

3. Select the *Generate PDF for printing* option, to publish the document in a printable, black and white format.

   This option generates a printable PDF document. The printed version appears in black and white (monochrome output with white background and black drawings and text) PDF document for printing purpose. The PDF document published in this manner is printed with default setup options. If you select the Generate PDF for printing option, the PDF

Option dialog box which opens by clicking the Setup options appears in read-only mode and the Advanced PDF page does not appear at all.

In the published PDF document, the *Descend* option available for descending into hierarchical blocks or the Attributes dialog box would not be available. In fact, such a PDF document would be an exact replica of the schematic.
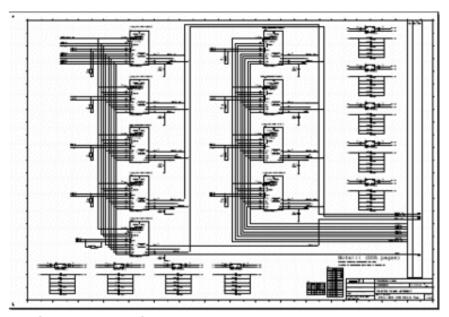


**Figure 2-7    Printable PDF document**

4. Specify the *Document Security* options to restrict access to the document.

The default security method is *No Security*.

   a. To remove all security restrictions, keep the default selection as *No Security*.

   b. To restrict access to the published PDF document, select *Password Security* from the Security Method drop-down list.

   The *Change Settings* button is activated.

   c. Click the *Change Settings* button.

The Password Security - Settings dialog box is displayed.



**Figure 2-8   Password Security Settings**

The security settings of a Design Entry HDL-generated PDF document are the same as those of an Adobe-generated PDF document. A PDF document has two types of passwords: a document open password and a permission password. When you set open document password, anyone who tries to open the document must type in the password.

When you set a permission password, only those users who have specified the correct permission password can change the security settings. If the PDF document includes both types of passwords, it can be opened with either of the two passwords. However, a user can set or restrict a feature only with the permission password.

**d.** Select the *Require a password to open the document* check box to make published PDF document password protected.

**e.** Specify a password to open the document in the *Document Open Password* text box.

The published PDF document would open only when the correct password is specified.

**f.** Specify the permissions on the published PDF document by selecting the *Use a password to restrict printing and editing of the document and its security settings* check box.

These settings let you specify the extent of permission granted to the users of the published PDF document.

❍ *Printing Allowed*— You can specify the kind of printing allowed on the published PDF document. The available options are:

- *None*

- *Low Resolution (100 dpi)*

- *High Resolution*

❍ Changes Allowed— You can specify the kind of changes you want to allow the users to make on the published PDF document. The changes allowed include:

- *None*

- *Inserting, deleting, and rotating pages*

- *Filling in form fields and signing existing signature fields*

- *Commenting, filling in form fields and signing existing signature fields*

- *Any except extracting pages*

**g.** Select the *Enable copying of text, images, and other content* check box to grant the corresponding permissions to the users of the published PDF document.

**h.** Select the *Enable text access for screen reader devices for the visually impaired to* o grant the corresponding permissions to the users of the published PDF document.

**5.** To make changes to the PDF setup, click the *Setup* button. It launches the PDF Setup dialog box, which is effectively the same as the Design Entry HDL Options— PDF Option dialog box.

**6.** Specify the location and the file name in the Save Dialog Box to save the published PDF document.

A progress dialog appears showing the progress of the PDF document being published. After successful generation of the PDF document, the following message appears which prompts you to view the published PDF document.
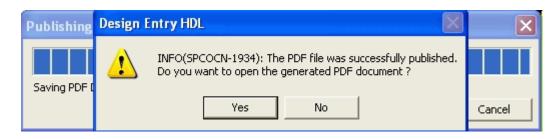


**Figure 2-9   PDF successfully generated**

**Note:** Similarly, you can publish PDF documents from the `schcref` and `schvar` views.

**Note:** Grids are not visible in the published PDF documents.

*Important*

If you find that the alignment of some text items is disrupted in the published PDF document, set the PDFFONT environment variable to a value ranging between 1 and 4. This environment variable stretches the height and width of a text object to align it properly as it appears in the actual design. Republishing a design after setting this variables fixes the text alignment issues.

# Publishing a PDF Document from the Console

You can also publish a PDF document from Design Entry HDL console using the `publishpdf` command.

**Syntax**

```
publishpdf [-all]|[-range <pages from>, <pages to>]|[-active] [-
print] -file
```

where,

| Parameter | Description |
| --- | --- |
| `-all` | Publishes all pages present in current design. |

| Parameter | Description |
|---|---|
| `-range` | Publishes a specified range of pages. |
| `-active` | Publishes only the current page. |
| `-print` | Publishes a printable black and white PDF. |
| `-file` | Specifies a file name with path to save the generated PDF document. This is a mandatory parameter. |
| `-version` | Displays the current version number of the Design Publisher solution. |

### Examples

```
publishpdf -all -file c:\proj1\sch1.pdf

publishpdf -range 1,10 -print -file c:\proj1\sch2.pdf
```

# Publishing a PDF Document from the Command Line

Similarly, you can publish a PDF document from the command line using the `publishpdf` command.

### Syntax

```
publishpdf -proj <project_name> [-cdslib] [-lib] [-cell] [-view] [-
page] [-all] [-range] [-active] [-print] [-extscrpt][-variant
<variant name>]-file <destination_path> [-product] [-noprogress]
```

where,

| Parameter | Description |
|---|---|
| `-proj` | specifies the name of the source project's `.cpm` file |
| `-cdslib` | specifies the `cds.lib` file |
| `-lib` | specifies the library name |
| `-cell` | specifies the cell name |
| `-view` | specifies the view name |

| Parameter | Description |
|---|---|
| `-page` | specifies the page name |
| `-all` | publishes all the pages present in the current design |
| `-range` | publishes a specified range of pages. If more than one page parameters are specified, the parameter specified last is honored.<br><br>In the example,<br><br>`publishpdf -proj test_proj -all -range 1,6 -file d:\project\proj1.pdf`<br><br>the winning parameter is `-range` |
| `-active` | publishes only the current page |
| `-print` | publishes a printable black and white PDF |
| `-file` | specifies a file name with path to save the generated PDF document. This is a mandatory parameter. |
| `-extscrpt` | specifies the path of the xml file which contains javascript to insert in PDF. |
| `-product` | specifies the product license. |
| `-variant` | specifies a variant for which the PDF is to be published<br><br>`-variant <variant name>` |
| `-noprogress` | Does not print dots indicating the progress of the publishpdf operation to the console. |

**Examples**

```
publishpdf -proj project1.cpm -all -file c:\project1\project1.pdf

publishpdf -proj project2.cpm -range 1,10 -print -file
c:\project2\project2.pdf

publishpdf -proj project2.cpm -range 1,10 -print -file
c:\project2\project2.pdf -noprogress
```

**Note:** Any errors during PDF generation are saved in a `pdf.log` file.

*Tip*

> You can also access help on the syntax of the `publishpdf` command by running the following command at the command prompt:
>
> ```
> publishpdf -help
> ```

## Specifying Options for Page Setup from the Command line

You can configure the page setup options using the command line switches also.

**Syntax**

```
publishpdf -proj <project_name> [-w] [-h] [-unit] [-lmargin] [-rmargin] [-tmargin] [-bmargin] [-scale] [-fitpage] [-landscape]-file <destination_path>
```

where,

| Parameter | Description |
|---|---|
| -w | specifies the width of the page |
| -h | specifies the height of the page. |
| | **Note:** A value of 0 for *w* and *h* represents *auto-size*. In such a case, page margin can only be controlled by using the *-lmargin* parameter. |
| | For auto-size, inches is considered as the standard unit. Therefore, you need to ensure that the value of *-lmargin* is expressed in inches. |
| -unit | specifies the unit as either inches or millimeter. |
| -lmargin | specifies the left margin of the page in the specified unit. |
| -rmargin | specifies the right margin of the page in the specified unit. |
| -tmargin | specifies the top margin of the page in the specified unit. |
| -bmargin | specifies the bottom margin of the page in the specified unit. |
| -scale | specifies the scaling factor (in percentage) by which the page is to be scaled (reduced or enlarged) |

| Parameter | Description |
|-----------|-------------|
| -fitpage | specifies that the document should be scaled to fit on the page. With this switch, scale is ignored. |
| -landscape | specifies that the page orientation is landscape. By default, the orientation is portrait. |

## Examples

```
publishpdf -proj project2.cpm -w 8.5 -h 11 -lmargin 1.2 -rmargin 1.2
-scale 80 -landscape c:\project2\project2.pdf
```

```
publishpdf -proj project1.cpm -tmargin 1 -bmargin 1 -fitpage
c:\project2\project1.pdf
```

# 3

# Exporting Physical or Packaged Designs to PDF

The Allegro Design Publisher solution (referred to as the *Publish PDF* utility in this user guide) facilitates viewing of complex designs made in Allegro PCB Editor or in Allegro Package Designer in a Portable Document Format (PDF) file. The design is made viewable independent of the application. This feature is especially useful when you need to share physical designs with experts who might not have Allegro PCB editor installed on their systems.

The pdf interface for physical design uses the book model where each Allegro artwork film is rendered as a separate page.

*Important*

> The *Publish PDF* utility is available on Windows, Sun Solaris, and Linux platforms. You need to have the PA1220 (Allegro Design Publisher XL) license installed to benefit from this utility. In addition, this utility and the corresponding command are not available with the Cadence SiP Digital SI and Cadence SiP Digital Architect licenses.

## Layers and Objects on the PDF

Layers are used to filter out visible information on a PDF document. Objects in a design are stored in predefined layers. You can control the display of various objects, such as components, pins, vias, and nets, in the published PDF document by showing or hiding specific layers. Board objects are categorized in 13 layers as mentioned below:

**Note:** Wire bonds objects are not exported to PDF by default. You can, however, export wire bonds as clines by adding the WIRE subclass to the Artwork Control Form. To do so, choose *Manufacture – Artwork*. In the Artwork Control Form, under available films, right-click on any of the listed subclasses and choose *Add*. In the Subclass Selection dialog box, check *WIRE* under *CONDUCTOR*.

■    Page Header/ Footer

■	Board Outline

■	Components

■	Nets

■	Test Points

■	Text

■	Route Keepin

■	Package Keepin

■	Route Keepouts

■	Package Keepouts

■	Via Keepouts

■	Drawing Origin

■	All Others

Each layer represents information on the corresponding object of the physical design. For example, the *Page Header/Footer* layer represents the page border and the *Nets* layer represents nets.

# Publishing a Physical or Package Design as a PDF

The pdf interface for physical design uses the book model where each Allegro artwork film is rendered as a separate page. After setting up the required artwork films, you can publish the physical or package design as a PDF.

For information of setting up Artwork, see *Chapter 6: Generating Artwork* in *Allegro User Guide: Preparing Manufacturing Data*.

You can create the PDF using the UI options or from the command line.

■	Publishing a Physical Design from the UI

■	Publishing a PDF Document from the Command Line

／Important

>	Before you start using the Publish PDF utility, ensure that you have Adobe Reader version 7.0 or later installed on your system.
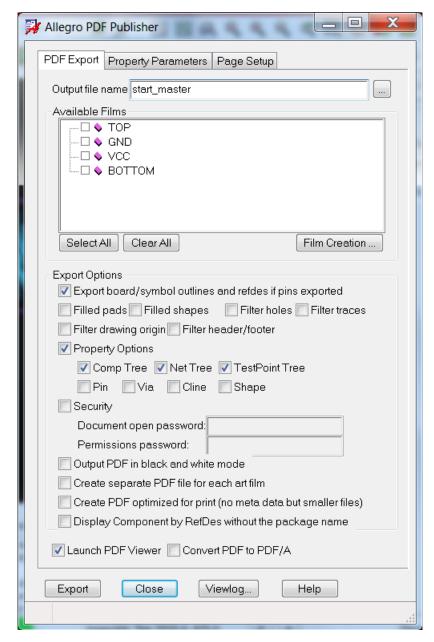
**Note:** You must create the required set of films using the artwork command before creating PDF.

## Publishing a Physical Design from the UI

To publish a physical design as a PDF document, perform the following steps:

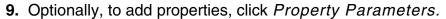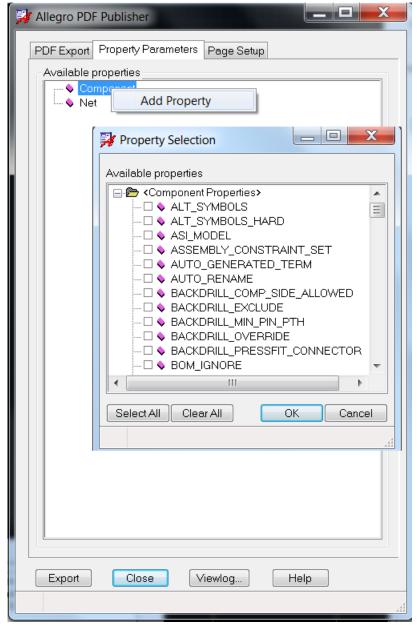**1.** Choose *File – Export – PDF*. The Allegro PDF Publisher dialog box displays.



> **Note:** You can also use the console command <u>pdf out</u> to display the Allegro PDF Publisher dialog box.

**2.** In the *Output file name* field specify the PDF file name.

**3.** From the *Available Films* list, select the art films to export.

**4.** Alternatively, use *Select All* and *Clear All* to add or remove the selection.

**5.** Click *Film Creation* button to invoke *Artwork Control Form*.

**6.** Specify the *Export Options*. Use these options to customize the PDF file.

For details on all the Export Options, see <u>pdf_out</u>.

**7.** Select the checkbox *Launch PDF Viewer* to open the PDF file after export is completed.

**8.** Select the checkbox *Convert PDF to PDF/A* to create PDF for archiving purpose.

**9.** Optionally, to add properties, click *Property Parameters.*



**a.** Click the *Property Parameters* tab.

**b.** Right-click and choose *Add Property*. The *Property Selection* dialog box displays.

**c.** Select the properties to add and click *OK*.

**d.** To select all the properties click *Select All*.

**e.** To deselect all the properties click *Clear All*.

**Note:** Note that you can export properties for nets and components to a PDF.

**10.** Optionally, to specify Page settings:
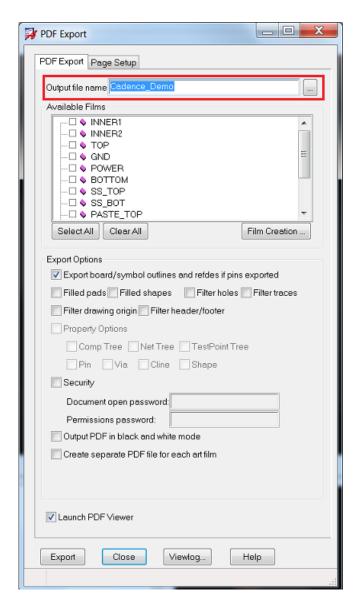


    **a.** Click the *Page Setup* tab.

    **b.** Specify the units, paper size, orientation, margins, and scaling.

    **c.** Specify text size for header/footer.

**11.** Click *Export* to export the PDF. The design is exported as PDF.

## Exporting PDF Without License

The *Publish PDF* utility can also be used with minimum features in the absence of the necessary license. You can generate a basic PDF of the design that does not contain tree structures and metadata.
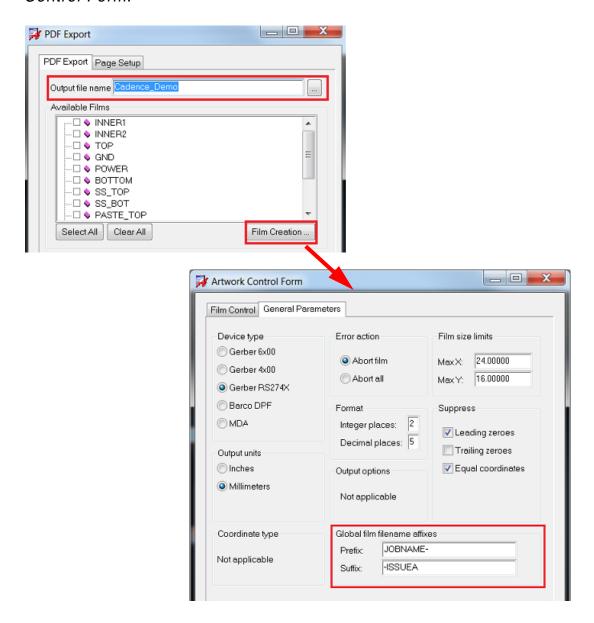
When exporting a basic PDF from PCB Editor, the *Conversion to PDF/A* option is not available.

When the required license is not found, the user-interface of the *Publish PDF* utility changes as shown in the following image:

The name of the output PDF file is combination of two elements:

■  *Output file name* specified in the *PDF Export* dialog box

■  *Prefix* and *Suffix* specified in the of *Global film filename* section of the *Artwork Control Form*.

## Publishing a PDF Document from the Command Line

You can also publish a PDF document from operating system command line using the
`pdf_out` command.

**Note:** If the necessary license is not available, the command creates a basic PDF. A warning
message appears for features that are not supported.

## Syntax

```
pdf_out <design_name > [-s] [-l] [-B][-C] [-r] [-h] [-p] [-P] [-t] [-
U] [-n] [-m] [-i] [-v] [-e] [-S] [x] [-f <art_film_name1> -f <
art_film_name2>....…] [-c <config_file_name>] [-o output_name] [-u
user_pass] [-w perm_pass]
```

where,

| Parameter | Description |
|---|---|
| design_name | Mandatory parameter to specify the file name of the design to be exported. |
| -s | Creates a separate PDF file for each art film. By default a single PDF file is generated for all exported art films. The output file name for each film is: <br><br> `<output_file_name>_<film_name>.pdf` <br><br> or <br><br> `<prefix>_<output_file_name>_<film_name>_<suffix>.pdf` |
| -l | Lists all the artwork films in the physical design. |
| -B | Creates a PDF file in Black & White. By default, design colors are used in the PDF. |
| -r | Exports board outline, symbol outline, and refdes as well if pin is exported. |
| -P | Creates a size optimized PDF file for printing. No design data is exported. |
| -p | Fills pads on export. |

| Parameter | Description |
|---|---|
| -t | Excludes any traces or connection lines. |
| -U | Clears any filled spaces. |
| -f | Specifies the Art film name to export. By default, all art fils are exported. |
| -c | Specifies the configuration file used to control paper size, margins, scale factor, and export extra properties for Component and Net. |
| -o | Specifies the output file name. The default value is `<design_name>.pdf`.<br><br>**Note:** If the global filename affixes are specified,  the resultant output name will be: `<prefix>_<output_file_name>_<suffix>.pdf` |
| -C | Displays component by RefDes without package name. |
| -u | Specifies the user password to open the PDF file. |
| -w | Specifies the password to open and edit the PDF file. |
| -h | Excludes hole symbols. |
| -a | Excludes header/footer details. |
| -g | Excludes drawing origin. |
| -D | Creates PDF/A file. |
| -n | Exports metadata for Net. Displays a warning message if license not found. |
| -m | Exports metadata for Component. Displays a warning message if license not found. |
| -i | Exports metadata for Pin. Displays a warning message if license not found. |
| -e | Exports metadata for Cline. Displays a warning message if license not found. |
| -T | Exports metadata for TestPoint tee and TestPoint outline. Displays a warning message if license not found. |
| -v | Exports metadata for Via. Displays a warning message if license not found. |

| Parameter | Description |
|---|---|
| -S | Exports metadata for Shape. Displays a warning message if license not found. |

# Examples

Refer to the following examples to understand how to generate PDF files using `pdf_out` command line syntax.

### Example 1

Create a PDF file to export all art films, and export board outline, symbol outline, and refdes, if symbol pins are exported.

```
pdf_out test.brd -o test -r
```

### Example 2

Create a PDF file using a configuration file.

```
pdf out test.brd -o test -c pdf_out_config.txt
```

**Note:** The format of object property parameter is:

> `Object_Type/Property_Name`, one entry per line.

For example:

```
Component/IDF_OWNER
Net/DIFFP_PHASE_TOL_DYNAMIC
Net/MAX_VIA_COUNT
page_setup/unit=Inch or page_setup/unit=Millimeter
page_setup/paper_size_width=8.500
page_setup/paper_size_height=11.000
page_setup/margin_left=0.500
page_setup/margin_right=0.500
page_setup/margin_top=0.500
page_setup/margin_bottom=0.500
page_setup/scale_factor=fit_page or page_setup/scale_factor=1.50
```

**Example 3**

Create a PDF File to export two art films  TOP and BOTTOM.

```
pdf out test.brd -o test -f TOP -f BOTTOM
```

**Example 4**

Create separate PDF files for each of the exported art films.

```
pdf out test.brd -o test -f TOP -f BOTTOM -s
```

**Example 5**

Create PDF file to export the art films TOP and BOTTOM, and export metadata for component, net, pin, via, cline, shape:

```
pdf out test.brd -o test -f TOP -f BOTTOM -m -n -i -v -e -S
```

**Example 6**

Create basic PDF file without setting the necessary license to export the art film for BOTTOM layer and to export metadata for pin on this layer:

```
pdf_out test.brd -i -f BOTTOM
```

The output PDF file has a single page for BOTTOM layer and the following warning message is displayed at the command prompt:

```
Warning: Allegro PDF Publisher licensing is needed for 'Pin Property'. IGNORED.
```

*Tip*

> You can also access help for publishing PDF by running the following command at the command prompt:
>
> ```
> pdf out -help
> ```

# 4

# Navigating through the PDF

The Allegro Design Publisher solution (referred to as the *Publish PDF* utility in this user guide) facilitates viewing of complex designs made in Design Entry HDL schematic editor and Allegro PCB Editor in a Portable Document Format (PDF) file. The design is made viewable independent of the design application.

This feature is especially useful when you need to share complex schematic designs or physical design with experts who might not have design editors installed on their computers. You can also use the generated PDF for documentation and review purposes.

After publishing the PDF document, you can start browsing it. This section describes the structure of the published PDF document vis-à-vis Design Entry HDL schematic design or PCB Editor board.

When you open a published PDF for the first time, the following message box pops-up:
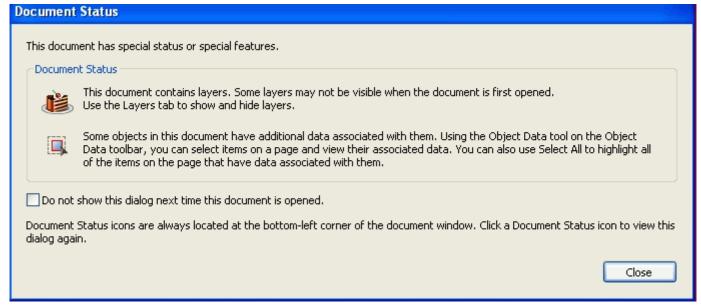


**Figure 4-1   Document Status Dialog Box of Adobe Reader 7**

47

The published PDF document contains the complete design information that you select to publish. The information is used for providing some navigation aids for navigating the design. This has been possible with the support for Object Data Tool in Adobe Acrobat Reader 7.0 or later.

The exported schematic information in the PDF document is displayed in the following tabs:

■    <u>Model Tree</u>

■    <u>Bookmarks</u>

■    <u>Layers</u>

■    <u>Hyperlinks</u>

■    <u>Thumbnails</u>

# Model Tree

In the data model tree, the navigation is based on three categories, Components, Nets and Test Point. The Test Point category is only available for PCB Editor. Each of these categories have a tree structure and all the information is stored in that structure. The trees are arranged in such a manner that navigating through the design is very convenient.

*Important*

> The Model Tree is only available if you open the PDF document in Adobe Reader 9 or later.

The model tree is not displayed on the PDF page, by default. To display the model tree, perform the following steps:

1. Click  Object Data Tool  (the Object Data Tool icon) on the toolbar of Adobe Reader. Alternatively, you can choose *Tools – Object Data – Object Data Tool* from the main menu of Adobe Reader.

2. Click any object on the published PDF document.

The model tree is displayed.



**Figure 4-2  Model Tree in Schematic and Layout PDF**

## Navigating Components in the Model Tree

### Design Entry HDL

All the components used in the design are listed under the Component tree. The reference designators and the names of components are displayed in the tree. If the tree item is a block or an unpackaged instance, the reference designator is displayed as "?". If the component is packaged, the reference designator of the component is displayed.
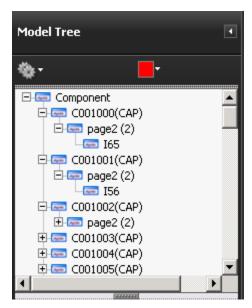


**Figure 4-3   Model Tree - Component List**

When you expand the component  tree a list of all the pages, on which the component is instantiated, is displayed. In case the component is a split part and has multiple symbols and these symbols are instantiated on different pages of schematic design, the list of all the pages on which this component symbols are present will be listed. On expanding the page number tree, the instance names of the components on that specific page are listed. The instance names are picked up from the `PATH` property assigned to the instance in Design Entry HDL.

➤   To navigate to an instance in the schematic representation on the PDF document, right-click the instance and select *Zoom to Selection* from the pop-up menu and navigate to the instance in the PDF document.

The following figure highlights an instance of the component `MICREL_SY58033UMG` which has the reference designator `CKM00B`.
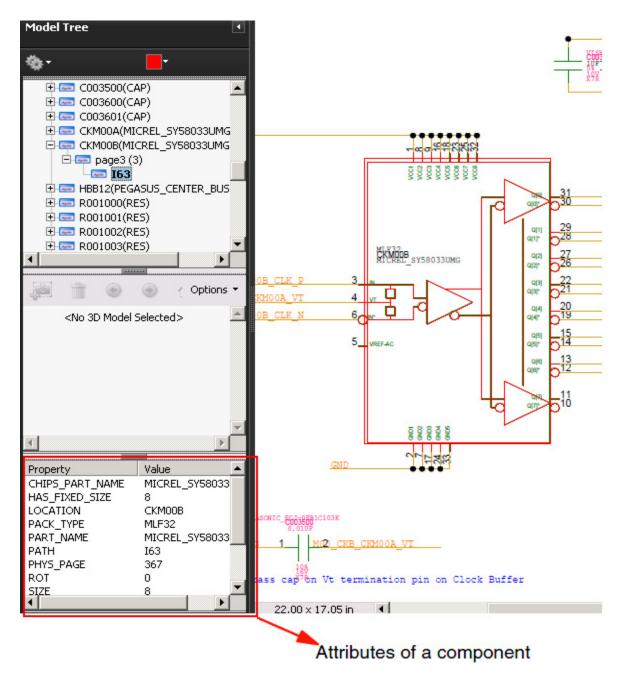
**Figure 4-5   Model Tree - Component Instance highlighted in the schematic PDF**

**PCB Editor**

All the components used in the design are listed under the Component tree. The reference designators and the names of components are displayed in the tree.
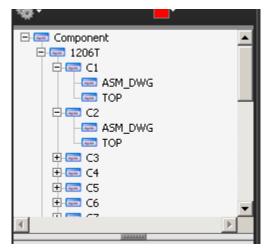


**Figure 4-6   Model Tree - Component List**

When you expand the component tree a list of all the artwork layers, on which the component is instantiated, is displayed.On expanding the refdes tree, the names of the art work layers on which components lies are listed.

To navigate to an instance in the physical representation  on the PDF document, right-click the instance and select Zoom to Selection from the pop-up menu and navigate to the instance in the PDF document.

The following figure highlights an instance of the component `PLCC28` which has the reference designator `U6`.
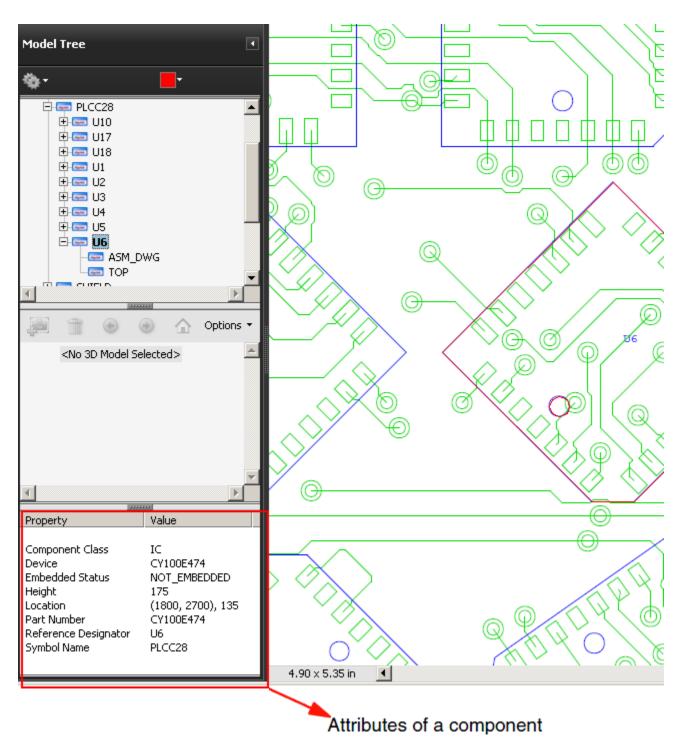
Figure 4-8  Model Tree -Component
Instance highlighted in the layout PD

**Note:** The Object Data Tool functionality of Acrobat Reader 7.0, lets you view the attributes of components, nets, and pins by clicking various areas of the published PDF document. When you select the object in the model tree, the PDF document populates the Attributes dialog box with the attributes of the object.

## Navigating Nets in the Model Tree

Similarly, you can navigate to specific nets from the model tree. All the nets in the design are listed under the Net tree according to their physical net names. On expanding the tree for each of the nets, the pages on which these nets are present are listed. You can right-click any of these pages and then choose *Zoom to Selection* from the pop-up menu, and navigate to the net in the PDF document.
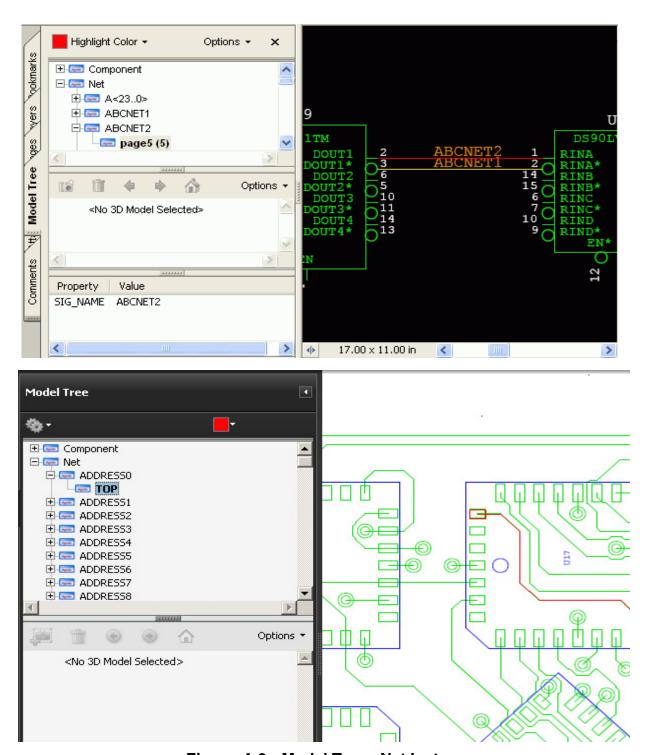
**Figure 4-9   Model Tree - Net Instance**

In case of buses, if the bus bits are tapped out, the individual bus bits or range of bus bits will also be visible under the complete bus tree. Under the individual bus bits or range of bus bits tree, the pages where they occur will also be listed.
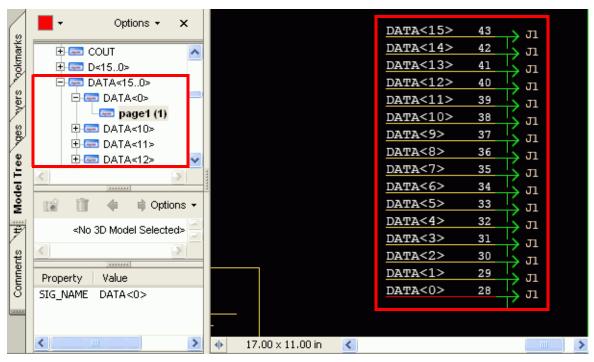


**Figure 4-10   Model Tree - Bus bit highlighted in the published PDF**

## Navigating Test point in Model Tree

The test point tree displays the net name the test point is assigned and test point layer (top or bottom). The test point data associated with the selected net is visible in the lower panel of the navigation pane. You can right-click any of these pages and then choose *Zoom to Selection* from the pop-up menu, and navigate to the test point in the PDF document. Test points are identified by a larger circle graphically drawn around the pad location for easy identification. You can select pins and vias identified as a test points, and view their attributes.
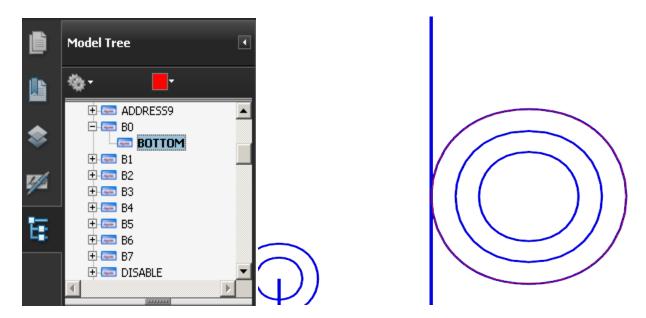
**Figure 4-11  Model tree- Test Point Instances**

# Bookmarks

Bookmark are used to quickly navigate through the PDF document.
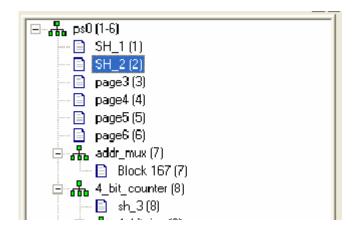
## Schematic PDFs

Bookmarks in a published PDF document display the hierarchy of the schematic design as captured in the document. Document hierarchy is shown in the Bookmarks panel of the PDF document in the form of a tree structure. This tree structure is the same as the hierarchy viewer tree in the schematic editor.

Similar to the display of sheet names in the Hierarchy Viewer of Design Entry HDL, the published PDF documents also display sheet names along with the block names and page numbers under each block in the design in the Bookmarks pane of the published PDF documents.

The pages appear just below the block, before the child blocks. Sheet names also appear in the Go To Sheet dialog box as well as the title bar of the main window.
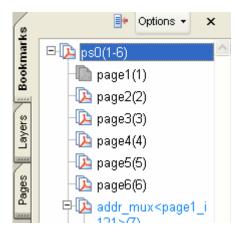


**Figure 4-12  Design Entry HDL Hierarchy Viewer and the Bookmarks panel displaying design hierarchy in the PDF document**

## Board PDFs

Bookmarks in a published PDF document display the exported artwork films of the physical design.
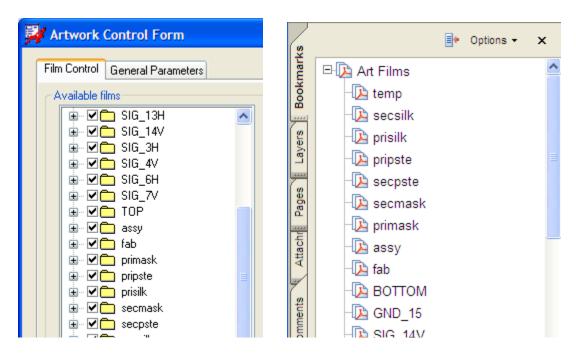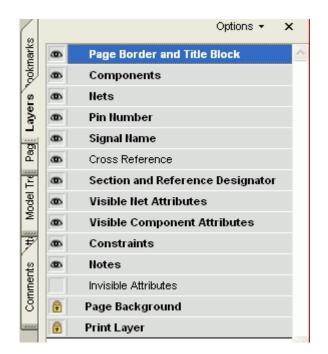


**Figure 4-13  Allegro PCB Editor Artwork films and the Bookmarks panel artwork films in the PDF document**

# Layers

Layers are used to filter out visible information on a PDF document. You use the eye icon to the left of a layer in the Layers panel to turn on or off the display of the corresponding object on the schematic drawing on the published PDF document.
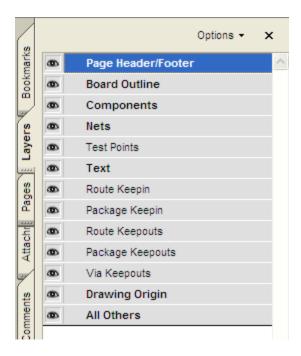


**Figure 4-14  The Layers panels in a schematic and board published as PDF**

Objects on a schematic are stored in predefined layers. You can control the display of various schematic objects, such as components, and nets, in the published PDF document by showing or hiding specific layers. Each layer represents information on the corresponding object of the schematic. For example, the *Page Border and Title Block* layer represents the page border and the title block and the *Nets* layer represents nets.

You can choose to export and make specific layers visible in the generated PDF document by selecting the *Export to File* and *Visibility* check boxes for the corresponding layer in the PDF Option page of the Design Entry HDL Options dialog box. The ability to turn layers on and off helps in increasing the readability of the schematic on the published PDF document.

# Hyperlinks

Use the hand pointer, and click on items to view details.
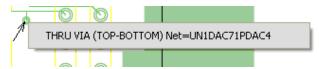
⚠ *Important*

> The pad object selection model is different then Allegro. To select an object you must always click on its left-most edge.
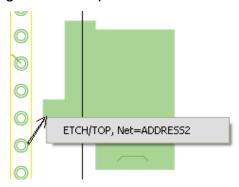
■   Via data

Click on a via on the page
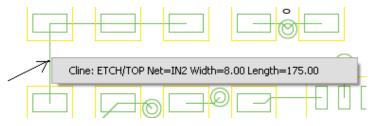


■   Shape

Click on the left-most edge of the shape.



■   Cline

Click the middle point of the cline if it contains one line segment; else click on a point that is close to the most left point.



■   Pin

Click on a pin to view the Pin details. Click Attributes, to view the pin properties in a pop-up window.
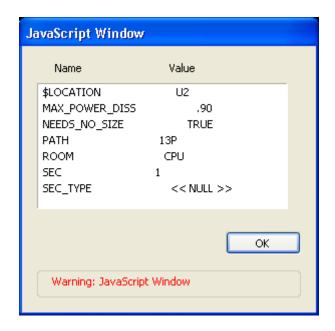


You can also view net and component properties in the Attributes dialog box in the published PDF document by selecting the *Attributes* menu command from the pop-up menu.

*Important*

The pop-up menu for displaying net attributes is visible only if you left-click the net name. It does not appear on clicking the net segment.

The components and net properties are displayed in the Attributes dialog box.
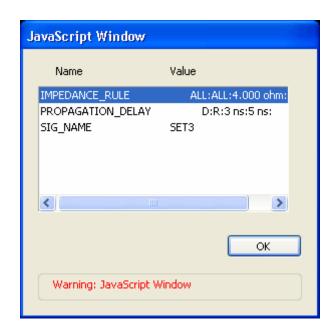


**Figure 4-15 Viewing Component and Net Properties**

**Note:** When you view properties in the Attributes dialog box of a component or a net, the following warning is displayed in red text at the bottom of the dialog box:

`Warning: JavaScript Window`

This warning is displayed by Adobe Acrobat to inform you that the dialog box is generated by Java Script and not by Acrobat.

**Note:** In Design Entry HDL, to view attributes on the bits of a bus, you need to specify indexes in the Attributes dialog box. This functionality is not available in the published PDF document. As a result, you cannot view attributes of a specific bus bit.

In a schematic PDF, hyperlinks provide additional support for:

■ Navigating through Hierarchical Symbols in schematic PDFs

■ Viewing Visible Component and Net Properties
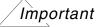
## Navigating through Hierarchical Symbols in schematic PDFs

You can click a component or a block and descend into the corresponding design or symbol. You can descend into a block or component by selecting the *Descend* menu option from the pop-up menu, which appears when you click the block or the component.



**Figure 4-16  The Descend menu option**
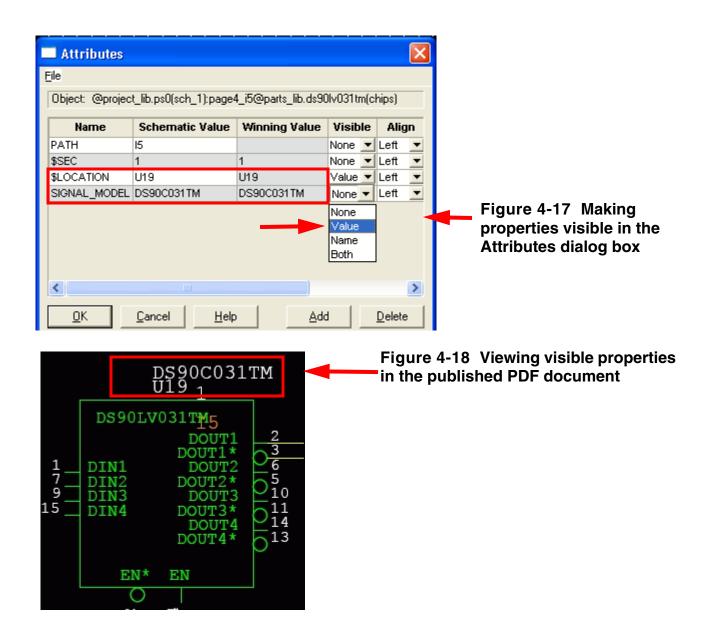
**Note:** In a published PDF document, a left mouse click performs certain functions, you would expect from a right mouse click. For example, customized pop-up menu to show component information is invoked by a left mouse click in a published PDF document.

△ *Important*

If you publish a page containing a hierarchical block without publishing its child pages, the *Descend* command will not descend down the hierarchy as the child pages are not available.

### Viewing Visible Component and Net Properties

You can view visible properties of nets and components in the published PDF document. Visible properties are the ones which have placeholders on the schematic.



**Figure 4-17  Making properties visible in the Attributes dialog box**



**Figure 4-18  Viewing visible properties in the published PDF document**

# Thumbnails

You can view the thumbnails of the pages by clicking on the *Pages* tab on the left panel of the published PDF document. Page thumbnails provide mini-previews of document pages.
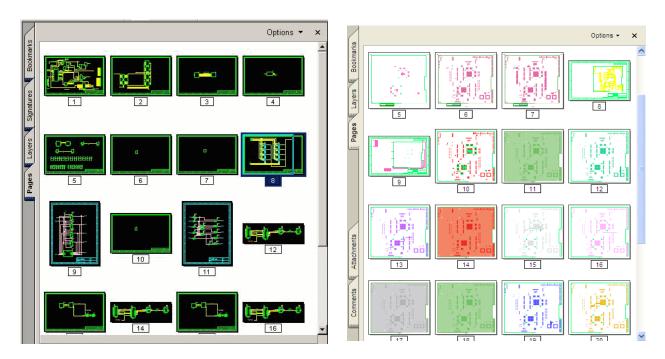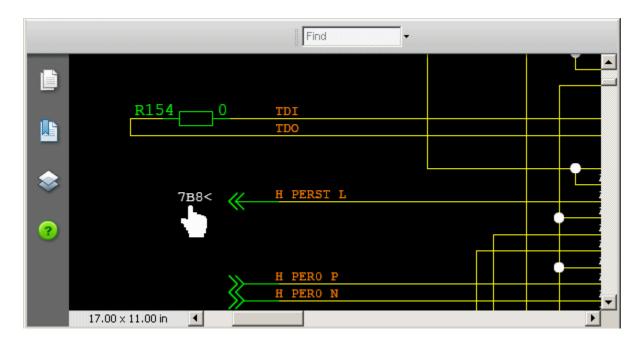
**Figure 4-19  Thumbnails of schematic and board PDFs**

# Cross-References as Links

In a schematic published as a PDF, you can click cross-references in a design to navigate to destination location. As you move the mouse pointer over a Cref link in the PDF document, it changes to a hand pointer.

# Index