



Importing Eagle Files

Copyright Notice

Copyright in the whole and every part of this software and manual belongs to RS Components and may not be used, sold, transferred, copied or reproduced in whole or in part in any manner or in any media to any person, without the prior written consent of RS Components. If you use this manual you do so at your own risk and on the understanding that neither RS Components nor associated companies shall be liable for any loss or damage of any kind.

RS Components does not warrant that the software package will function properly in every hardware software environment.

Although RS Components has tested the software and reviewed the documentation, RS Components makes no warranty or representation, either express or implied, with respect to this software or documentation, their quality, performance, merchantability, or fitness for a particular purpose. This software and documentation are licensed 'as is', and you the licensee, by making use thereof, are assuming the entire risk as to their quality and performance.

In no event will RS Components be liable for direct, indirect, special, incidental, or consequential damage arising out of the use or inability to use the software or documentation, even if advised of the possibility of such damages.

RS Components reserves the right to alter, modify, correct and upgrade our software programs and publications without notice and without incurring liability.

DesignSpark is a Trademark of RS Components, Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

Eagle is the copyright of CadSoft

All other trademarks are acknowledged to their respective owners.

Copyright © RS Components. 1997-2011. All rights reserved. E&OE

Issue date: 23/02/11 iss 1

RS Components Ltd
International Management Centre
8050 Oxford Business Park North
Oxford
OX4 2HW
United Kingdom
Tel: +44 (0)1865 204000
Fax: +44 (0)1865 207400

Contents

- CONTENTS 3
- CHAPTER 1. CONVERTING DESIGNS 4
 - Overview 4
 - Supported Versions of Eagle..... 4
 - Exporting Designs 4
 - Importing Eagle Designs into DesignSpark 6
 - Limitations on Design Transfer..... 6
 - PCB Spacings 6
 - Unfinished Tracks..... 6
- CHAPTER 2. CONVERTING LIBRARIES..... 7
 - Eagle Library Structure 7
 - Exporting Libraries 7
 - Importing Libraries into DesignSpark..... 9
 - Importing an Intermediate file to create Schematic Symbols 9
 - Importing an Intermediate file to create PCB Footprints12
 - Importing an Intermediate file to create Components15

Chapter 1. Converting Designs

Overview

Eagle designs (Schematic and PCB) can be imported into DesignSpark, as can symbol, footprint and component libraries.

Supported Versions of Eagle

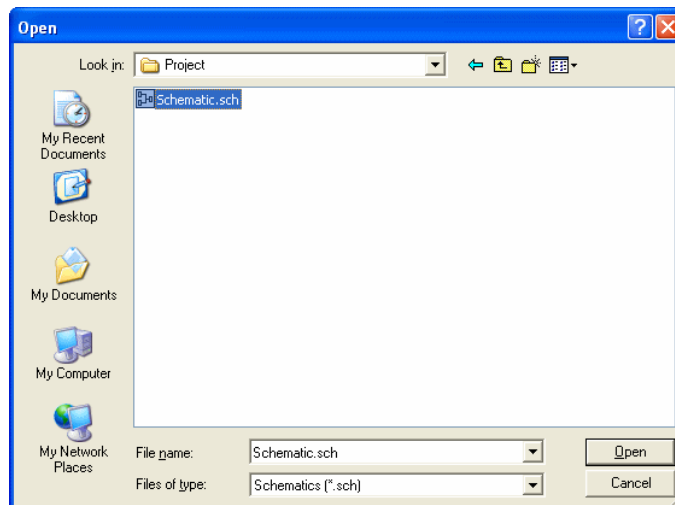
DesignSpark supports Eagle V5.x plus older versions back to V4.01 and V4.11. ULP files are supplied with DesignSpark and are used to export from Eagle format into an Intermediate ASCII format. These files are used as the import into DesignSpark by simply then opening them.

Exporting Designs

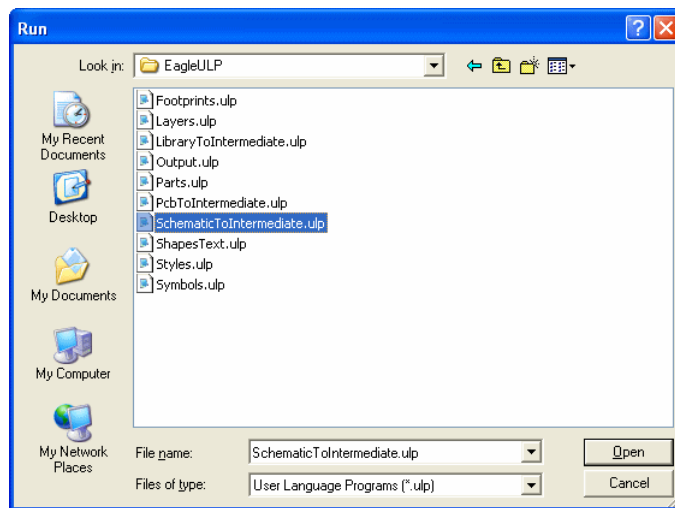
Both Schematic and PCB designs are exported in the same way from Eagle using the same menu options and ULP mechanism. The only difference is that one ULP file is run on Schematic design and the other run on the PCB design.

► To export an Eagle design into an ASCII file

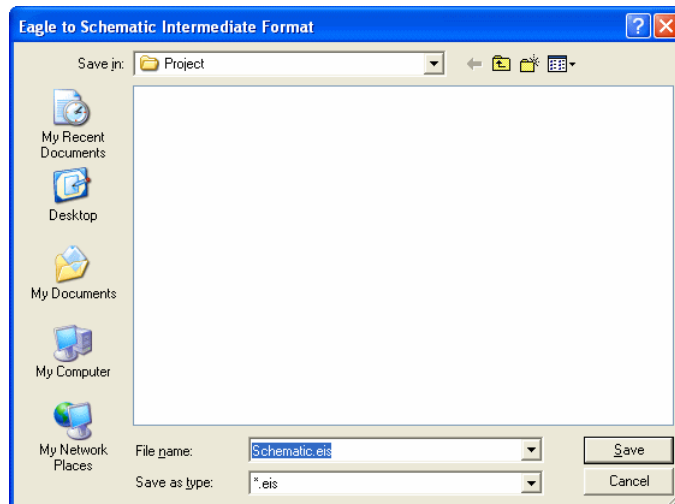
1. Start Eagle from the Task bar or from a desktop icon if you have created one.
2. From the **Project Manager**, select the **Open** option from the **File** menu.
3. Under **Open >**, choose **Schematic** or **Board** (PCB). This will depend on the design you wish to convert.
4. Locate the folder and file you wish to convert.



5. Select it and click **Open**.
6. The design will be displayed in the **Schematic** (or **Board**) editor.
7. You now need to convert the design to an Intermediate ASCII file format.
8. From the design editor toolbar, select the **ULP** icon (just under the Options menu item).
9. A **Run** window will open. From here, select the appropriate ULP file supplied for DesignSpark.
10. You will have to navigate away from the default Eagle ULP folder and locate the DesignSpark/EagleULP folder under which you will find the DesignSpark ULP files listed.
11. For a Schematic design, you will need to select the **SchematicToIntermediate.ulp** file.
12. If converting a PCB design, select the **PcbToIntermediate.ulp** file.



13. Once selected, click the **Open** button.
14. Choose the folder and file name of the intermediate file. By default, the folder and filename chosen will be for the folder and filename of the open Eagle design.



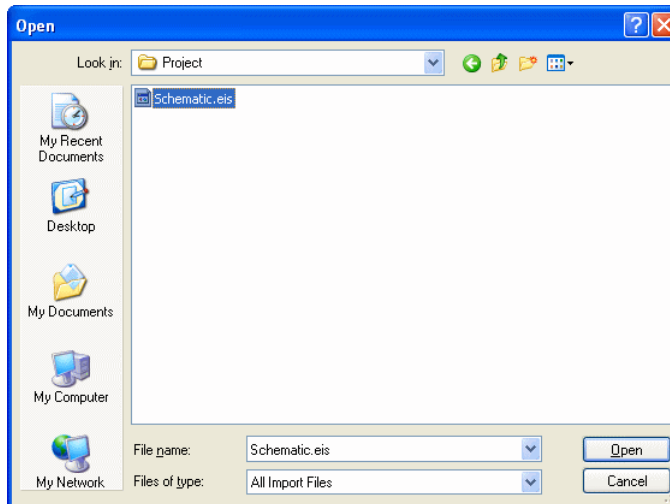
15. If you open and convert a Schematic design, the file extension will be .eis. If you opened a Board (PCB) file, the file extension will be .eip.
16. Click **Save**. The intermediate file (.eis or .eip) is written and will now be ready to import into DesignSpark.
17. Depending on the design size, the ULP file may take a few moments to run.
18. Watch the status bar at the bottom of the Eagle design, it will show you the progress as it runs. Once completed, the status changes to **Run: SchematicToIntermediate.ulp: finished**

Importing Eagle Designs into DesignSpark

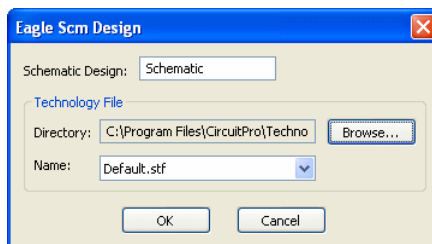
Now you must take the Intermediate ASCII file created in Eagle and import it into DesignSpark.

► To import an Eagle design into DesignSpark

1. Run **DesignSpark**
2. From the **File** menu, select **Import**
3. Locate the folder that contains the intermediate file and select it.



4. A small dialog will appear from which to add a file name and choose a Technology file. This dialog will be the same for a Schematic or PCB file except it will automatically reflect the detected design type.



5. Choose a file name or leave the name currently pre-set to the imported design name.
6. Unless you know what you're doing and have specific requirements, choose [None] as the technology file name. One is not mandatory for the conversion.
7. Click **OK** to import the design.
8. DesignSpark will redraw the design on the screen. This is now ready to use and edit as you wish.
9. Remember to save the design using the **Save** option from the **File** menu.

Limitations on Design Transfer

You should find that most designs should transfer to DesignSpark without any issues. However, there are a couple of limitations in Eagle to be aware of, as they may result in apparent errors in the corresponding DesignSpark PCB design.

PCB Spacings

The Spacing rules (clearances) defined in an Eagle PCB design are not accessible to the ULP script, so they cannot be transferred to DesignSpark. If you find that **Design Rule Check** in DesignSpark flags up many more errors than you would expect, you should check the settings in your **Spacings** in the **Design Technology** dialog. These should be edited to match the original settings from Eagle.

Unfinished Tracks

It is possible in Eagle to 'end' a track before it reaches the 'connect point' of a pad. On transfer to DesignSpark this kind of track will show up (on the screen and in **Design Rule Check**) as an '**unfinished track**'. This is not necessarily a problem, as the **Design Rule Check** will still check for completeness of the circuit as long as the track touches the pad, but you may again see more DRC errors than you might normally expect. These can be manually edited if you wish to remove them.

Chapter 2. Converting Libraries

Eagle Library Structure

Eagle library files (.lbr) contain Schematic symbols, PCB footprints and Part/Component information all in one self contained file.

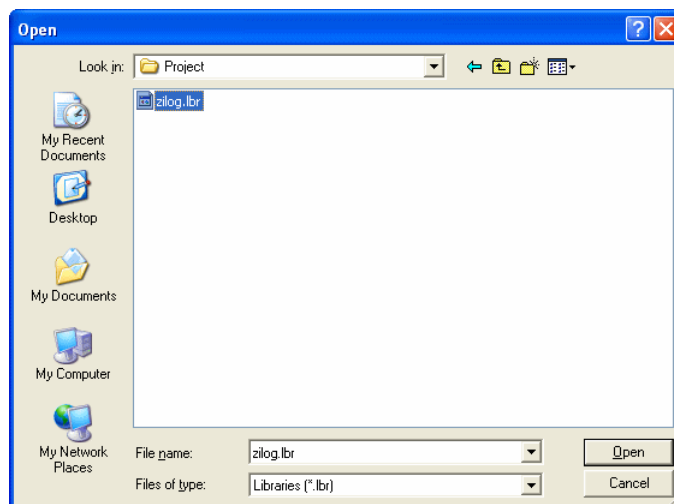
DesignSpark uses individual files for each library type Schematic symbols, PCB footprints and Part/Components. Therefore, during conversion, the one Eagle library file (and subsequent intermediate .eil file) is converted into separate DesignSpark files.

Exporting Libraries

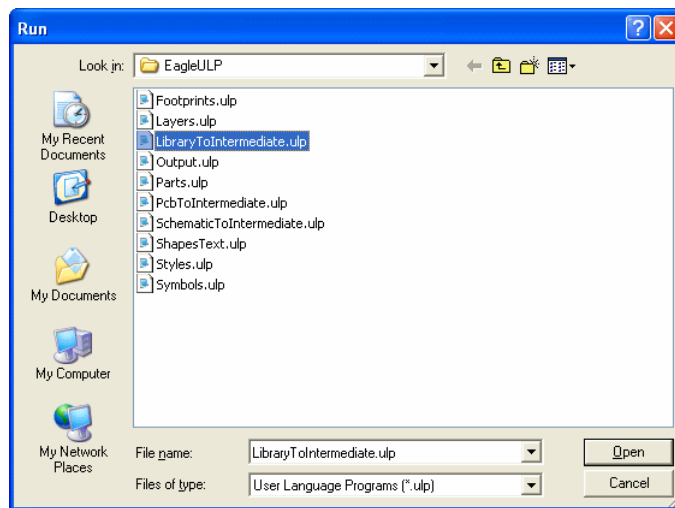
Each library required for conversion is exported using a ULP file to convert the library into an intermediate format file (.eil). There are not separate ULP files for the different library types, just the one.

► To export an Eagle library into an ASCII file

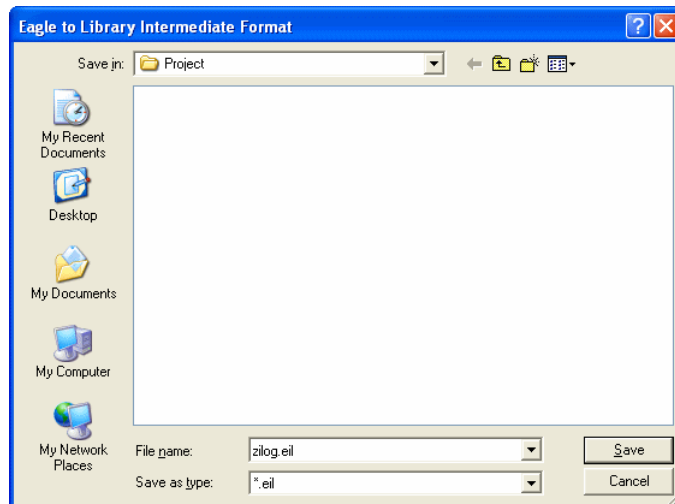
1. Start Eagle from the Task bar or from a desktop icon if you have created one.
2. From the **Project Manager**, select the **Open** option from the **File** menu.
3. Under **Open >**, choose **Library**.
4. Using the Open dialog, locate the folder and select the library to convert.



5. Click **Open**.
6. The library will open but nothing will be displayed at this point as this is the Library Manager from where you would edit and create the library entities.
7. You now need to convert the library to an Intermediate ASCII file format.
8. From the Library Editor toolbar, select the **ULP** icon (just under the Library and Options menu items).
9. A **Run** window will open. From here, select the appropriate ULP file supplied for DesignSpark.
10. You will have to navigate away from the default Eagle ULP folder and locate the DesignSpark/EagleULP folder under which you will find the DesignSpark ULP files listed.
11. Select the **LibraryToIntermediate.ulp** file.



12. Once selected, click the **Open** button.
13. Choose the folder and file name of the intermediate file. By default, the folder and filename chosen will be for the folder and filename of the open Eagle library.



14. It will request you select a .eil file.
15. Click **Save**. The intermediate file (.eil) is written and will now be ready to import into DesignSpark.
16. Depending on the library size, the ULP file may take a few moments to run.
17. Watch the status bar at the bottom of the Eagle design, it will show you the progress as it runs. Once completed, the status changes to **Run: LibraryToIntermediate.ulp: finished**

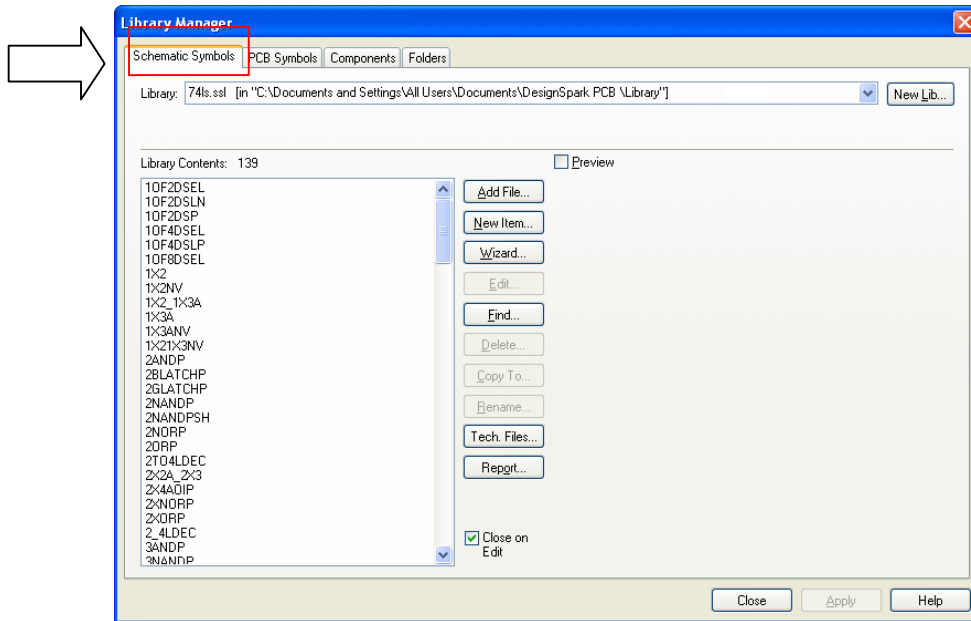
Importing Libraries into DesignSpark

As we said above, the one intermediate file for each Eagle library file must be read into three separate DesignSpark files to create the necessary library facets. To do this you must use the one .eil file and import it three times to each of the DesignSpark libraries (Schematic Symbol, PCB Footprint and Component). The procedure below details this for each type but it is actually the same for each.

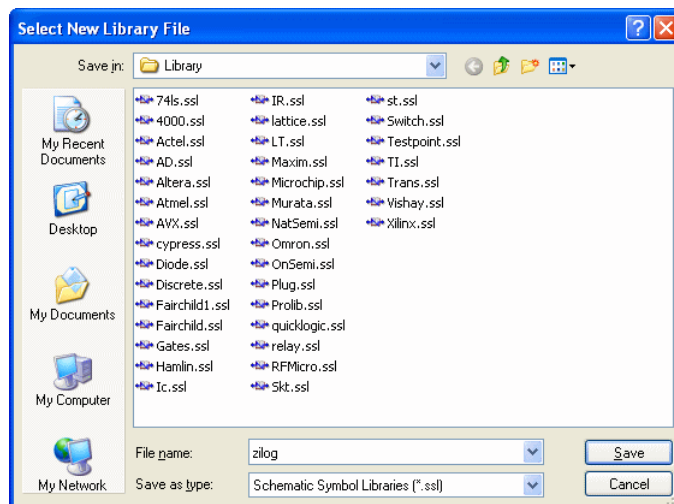
Importing an Intermediate file to create Schematic Symbols

► To import an Intermediate file to create Schematic Symbols

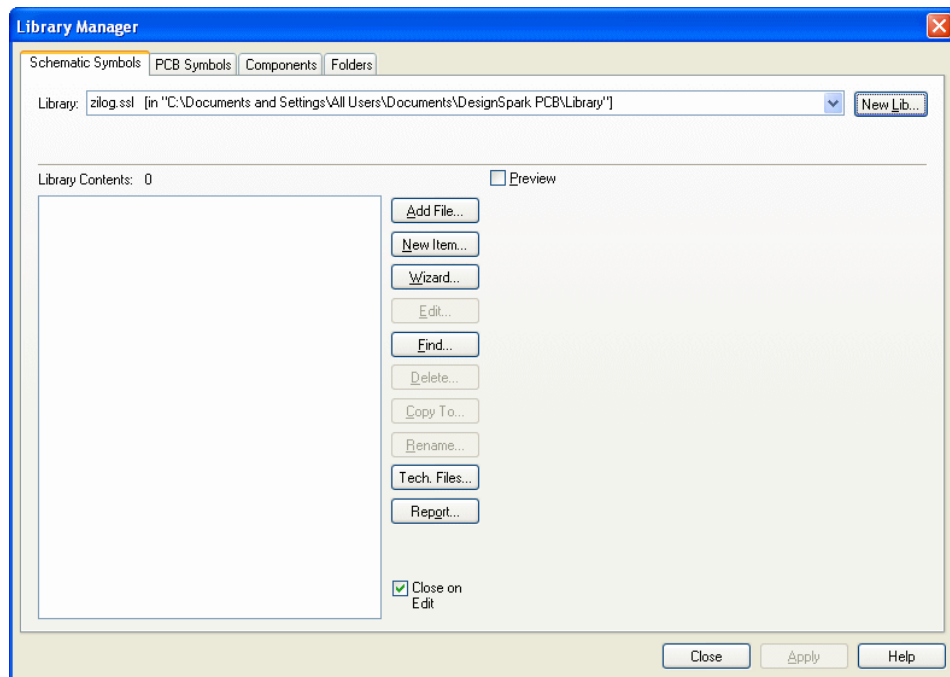
1. Run **DesignSpark**
2. From the **File** menu choose **Libraries**.
3. Click on the **Schematic Symbols** tab.



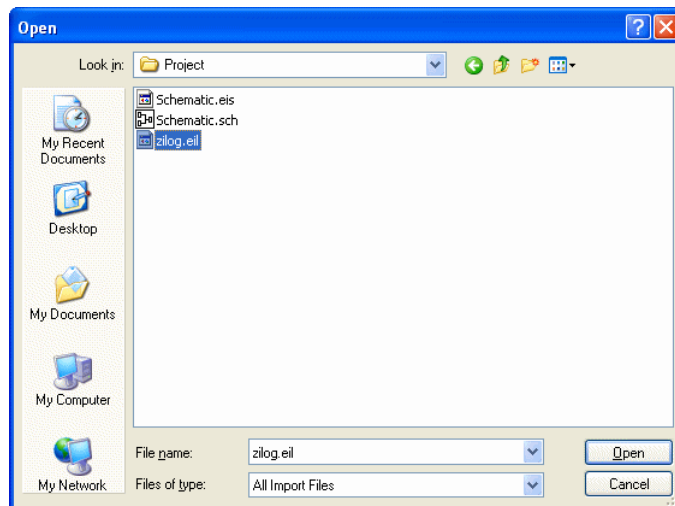
4. Click on the **New Lib** button. You will create a new library into which you will convert the Eagle library.



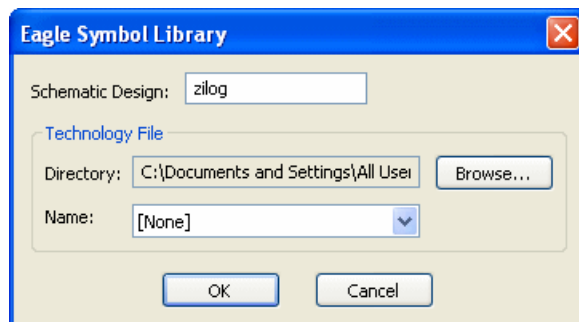
5. Type the name of the new library. You do not need to enter a file extension, it will automatically assign one for you.
6. Click the **Save** button.
7. The new empty library is displayed.



8. You will now convert and add the Eagle .ecl file.
9. Click the **Add File** button.

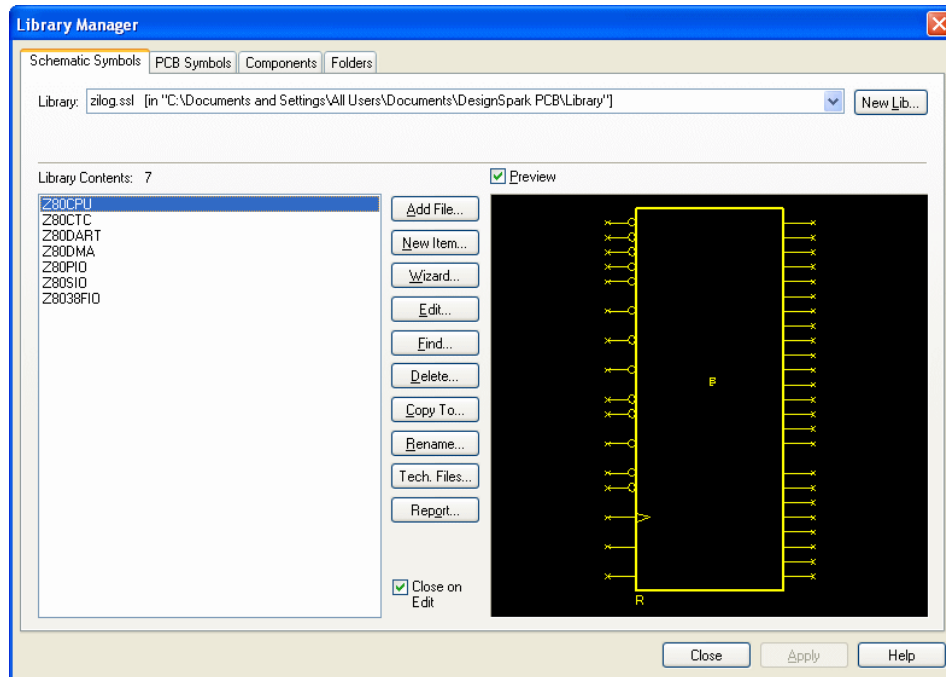


10. From the **Open** dialog, select the .ecl file to import. Don't forget, this file will be used three times to create the three corresponding libraries.
11. Press **Open**.



12. The import mechanism will automatically detect an Eagle library file.
13. The Eagle file is self-contained, you do not need to use a Technology file. You can use one if you wish to apply standardized names or colours.
14. Select the Technology file **Name** as **[None]**.
15. Press the **OK** button.
16. A progress bar will be displayed. Depending on how big the library being imported is, this may take a few moments to run.

17. Your library will open and be displayed in the **Library Contents** list.

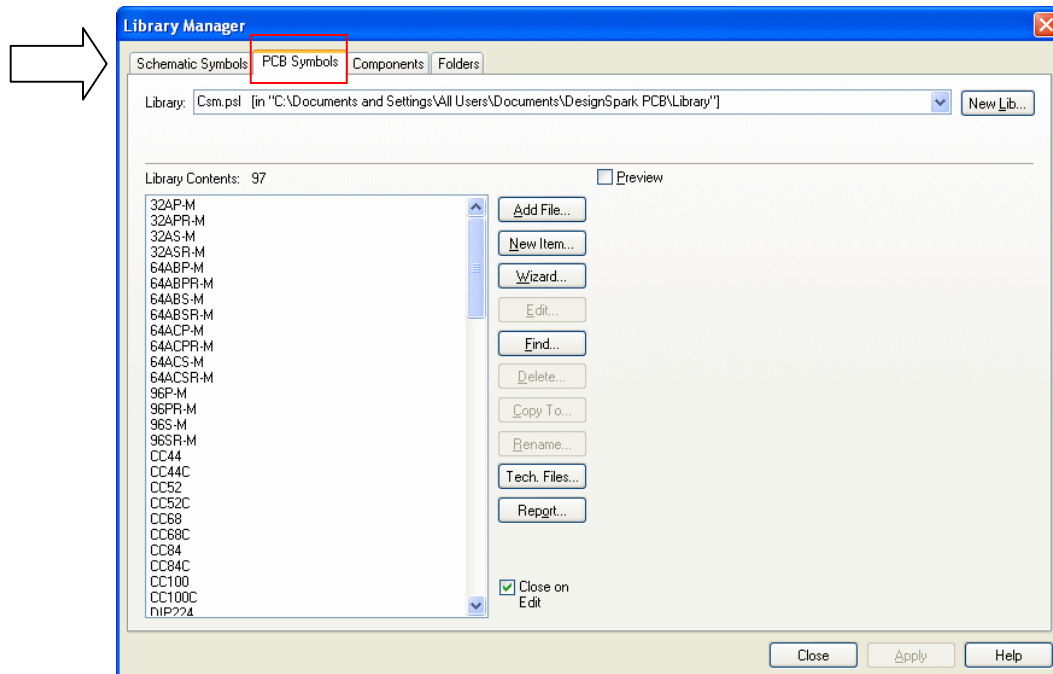


18. To run a quick visual check on the contents, select one of the items in the **Library Contents** list. Click the **Preview** button and the symbol will be displayed.
19. If you have more than one library to convert, repeat the procedure again from step 4.

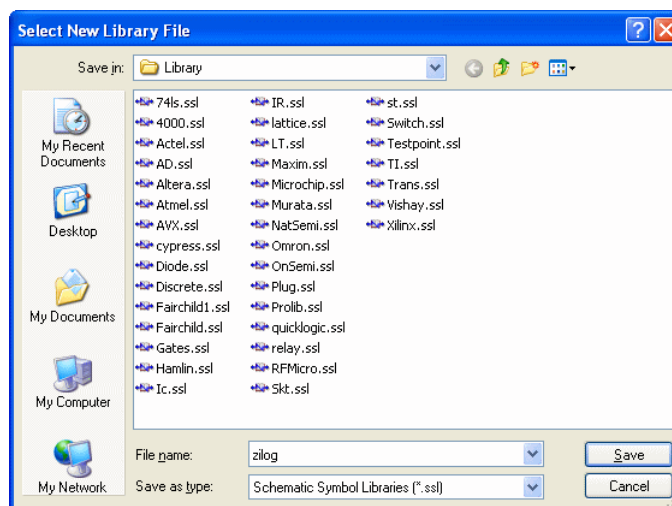
Importing an Intermediate file to create PCB Footprints

► To import an Intermediate file to create PCB Footprints

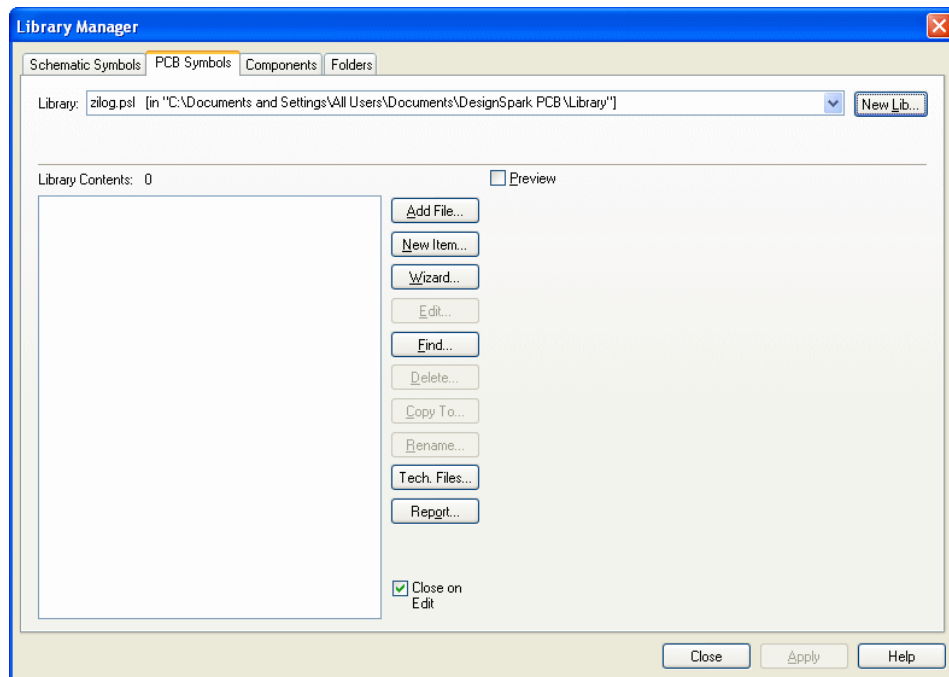
1. Run **DesignSpark**
2. From the **File** menu choose **Libraries**.
3. Click on the **PCB Symbols** tab.



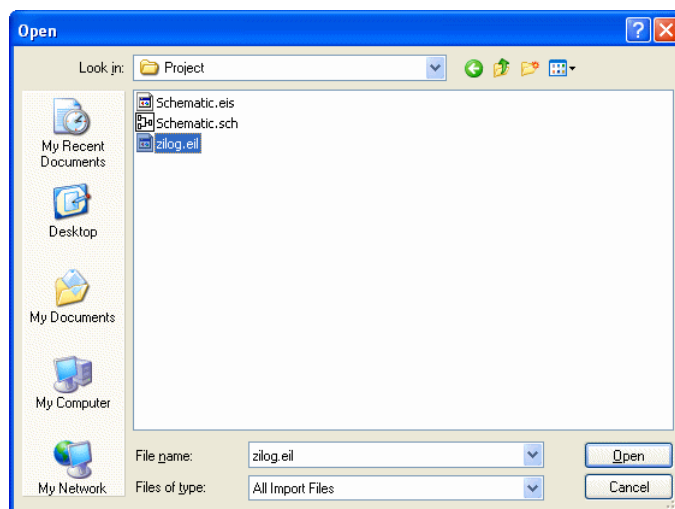
4. Click on the **New Lib** button. You will create a new library into which you will convert the Eagle library.



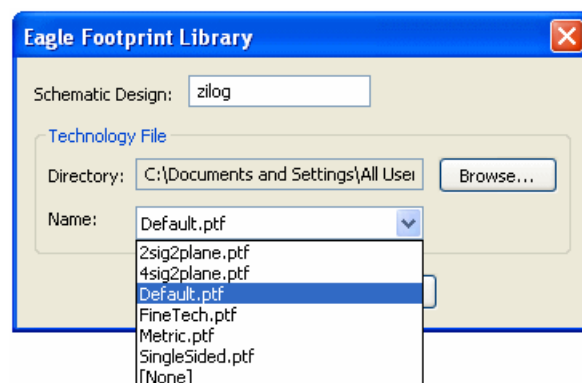
5. Type the name of the new library. You do not need to enter a file extension, it will automatically assign one for you. For a PCB Symbol library you may well add multiple intermediate files together to rationalise the footprints.
6. Click the **Save** button.
7. The new empty library is displayed.



8. You will now convert and add the Eagle .eil file.
9. Click the **Add File** button.

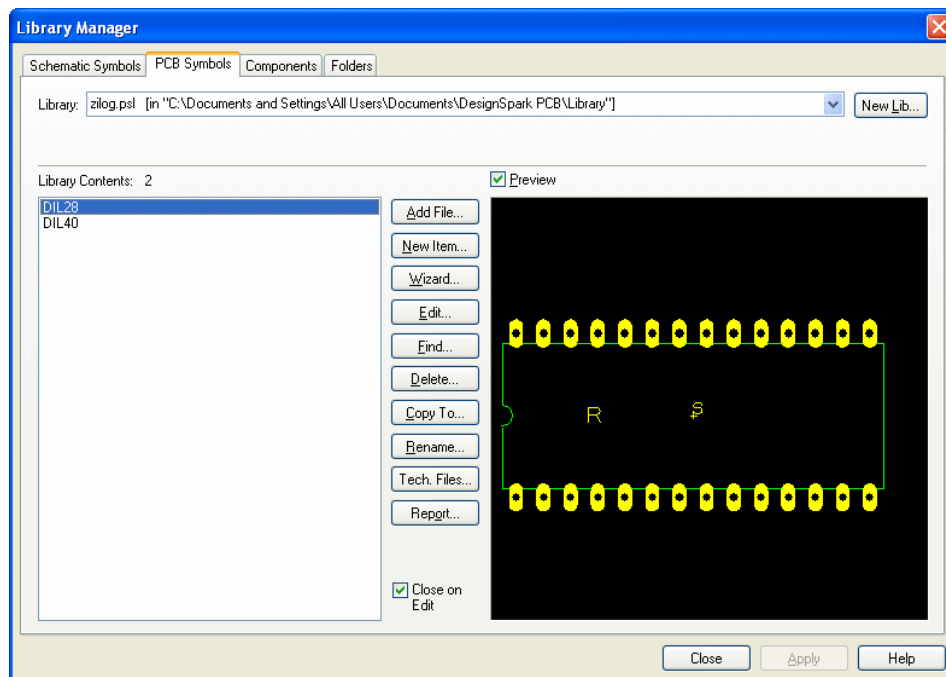


10. From the **Open** dialog, select the .eil file to import. Don't forget, this file has already been used, this if fine, it will be used one time more as well after this.
11. Press **Open**.



12. The import mechanism will automatically detect an Eagle library file.
13. The Eagle file is self-contained, you do not need to use a Technology file. You can use one if you wish to apply standardized names or colours.
14. Select the Technology file **Name** as **[None]**.
15. Press the **OK** button.

16. A progress bar will be displayed. Depending on how big the library being imported is, this may take a few moments to run.
17. Your library will open and be displayed in the **Library Contents** list.

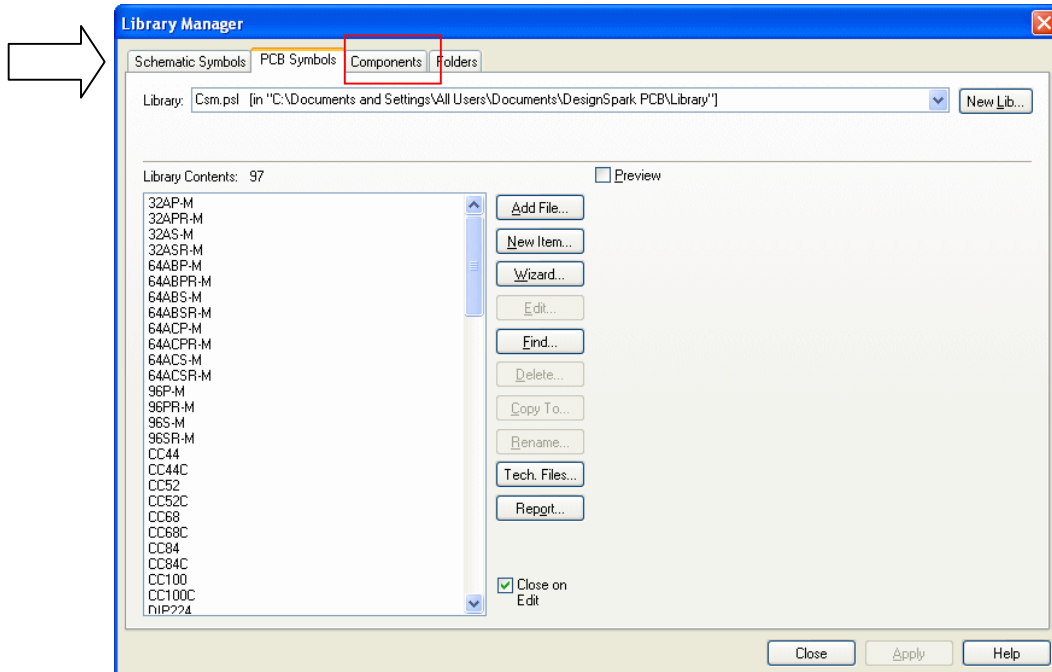


18. To run a quick visual check on the contents, select one of the items in the **Library Contents** list. Click the **Preview** button and the symbol will be displayed.
19. If you have more than one library to convert, repeat the procedure again from step 4. but this time, you may need to observe the note in step 5.

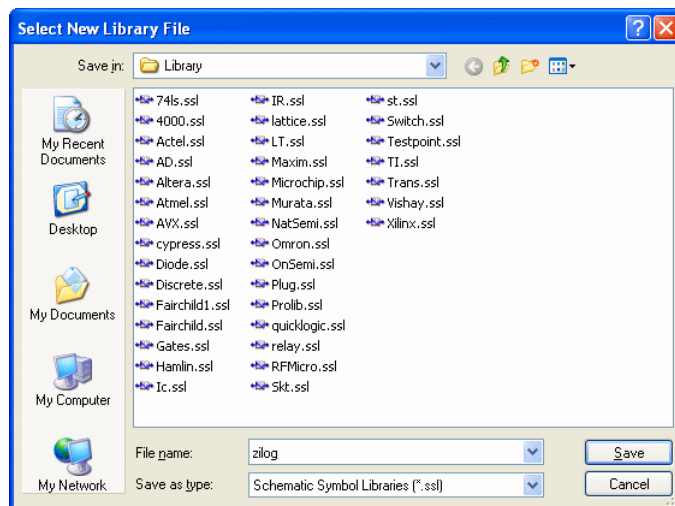
Importing an Intermediate file to create Components

► To import an Intermediate file to create Components

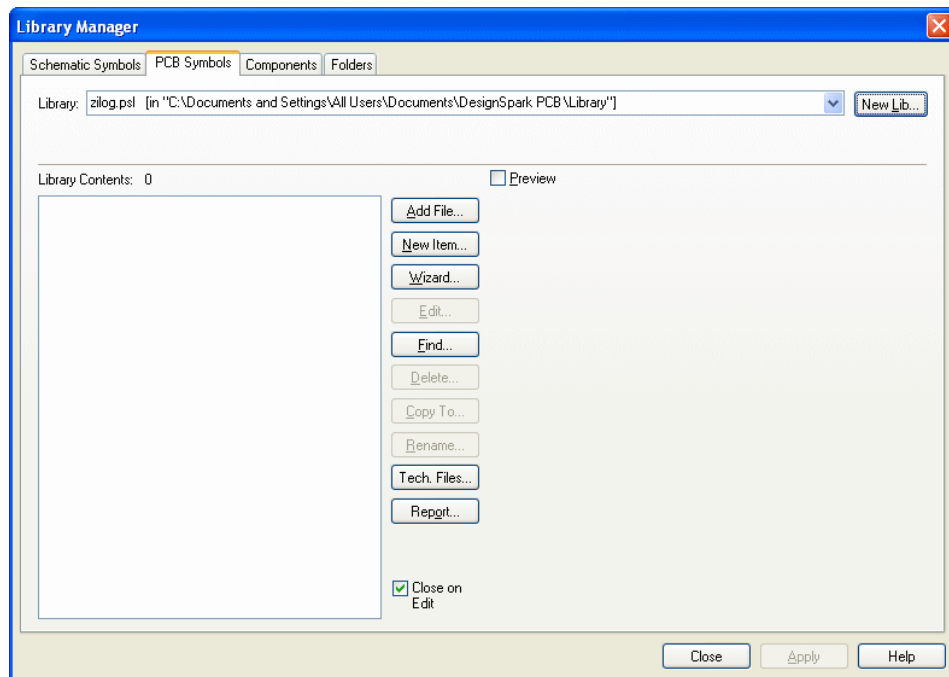
1. Run **DesignSpark**
2. From the **File** menu choose **Libraries**.
3. Click on the **Components** tab.



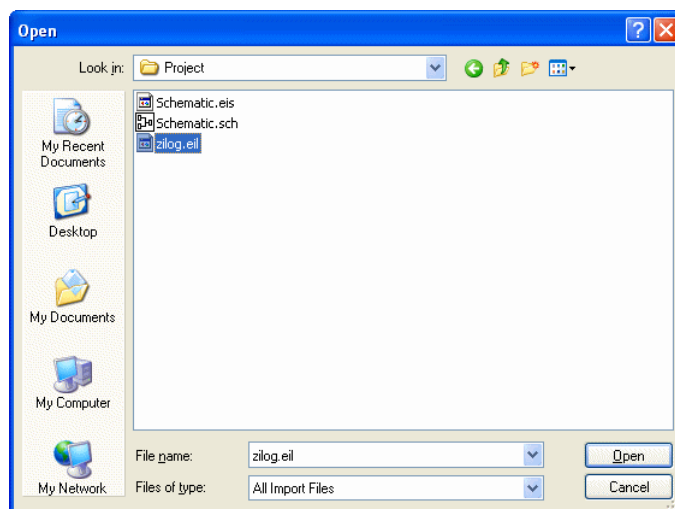
4. Click on the **New Lib** button. You will create a new library into which you will convert the Eagle library.



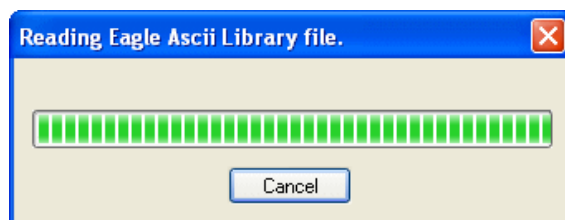
5. Type the name of the new library. You do not need to enter a file extension, it will automatically assign one for you.
6. Click the **Save** button.
7. The new empty library is displayed.



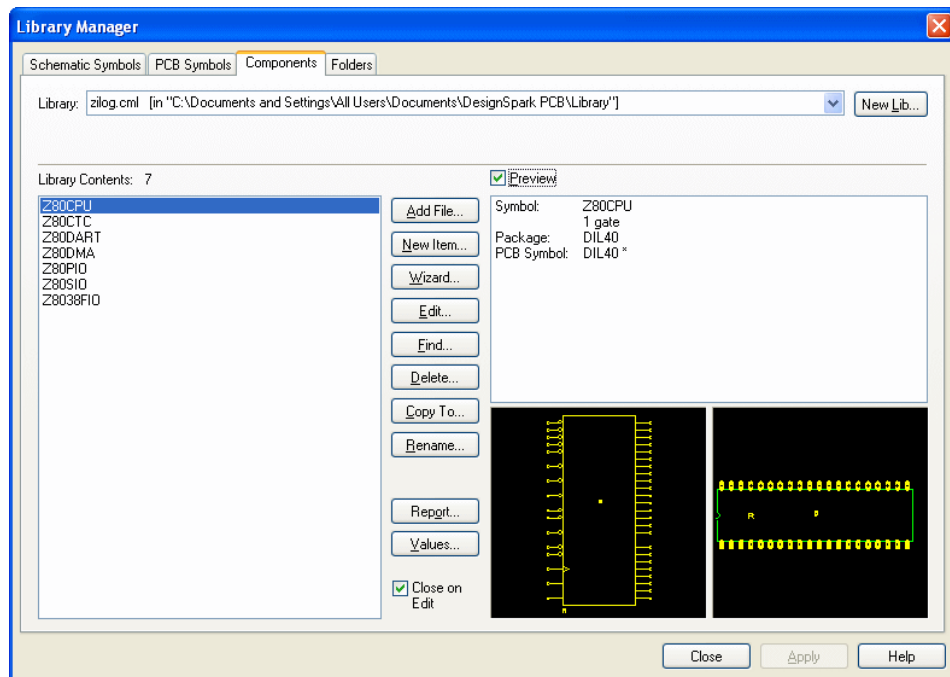
8. You will now convert and add the Eagle .eii file.
9. Click the **Add File** button.



10. From the **Open** dialog, select the same .eii file to import, this is one that has already been used.
11. Press **Open**.
12. The import mechanism will automatically detect an Eagle library file and launch straight into the importing.
13. A progress bar is displayed. Depending on how big the file is will depend on how long it is shown for, don't be surprised if you miss it.



14. A progress bar will be displayed. Depending on how big the library being imported is, this may take a few moments to run.
15. Your library will open and be displayed in the **Library Contents** list.



16. To run a quick visual check on the contents, select one of the items in the **Library Contents** list. Click the **Preview** button and the symbols plus the Component information will be displayed.
17. Check that the symbols are as expected and that the footprint is the correct type for the Component.
18. Importing the Component is the final operation to be done before you use it in either the Schematic or PCB designs.
19. If you have more than one library to convert, repeat the procedure again from step 4.