CircuitWorks 7
For SolidWorks

CircuitWorks 7 User's Guide

Issue 1 – 29th September 2004

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Introduction

CircuitWorks is designed to bridge the gap between SolidWorks and Electrical Computer-Aided Design (ECAD) systems by providing an intuitive tool that allows users to import and export Printed Circuit Board (PCB) designs between SolidWorks and ECAD systems.

Totally integrated within SolidWorks, CircuitWorks reads the industry standard IDF 2.0, IDF 3.0 and PADS file formats and produces accurate SolidWorks 3D assemblies of Circuitboards fully populated with their components.

Powerful tools within CircuitWorks allow the user to locate components in the assembly, edit reference designators, show changes to the assembly, and add annotations and much more.

SolidWorks assemblies can be saved as IDF files for use in the PCB design system. With its support for complex geometry, CircuitWorks is the ideal tool for getting complex Circuitboard profiles created in SolidWorks into a PCB design system accurately.

What are IDF Files?

IDF stands for Intermediate Data Format. It is a syntax developed specifically to provide a neutral format for exchanging critical PCB data between mechanical design, PCB layout, and physical design analysis systems.

IDF was jointly developed by SDRC and Mentor Graphics, and was released as IDF 2.0 in January 1992. The enhanced version 3.0 was released in October 1996.

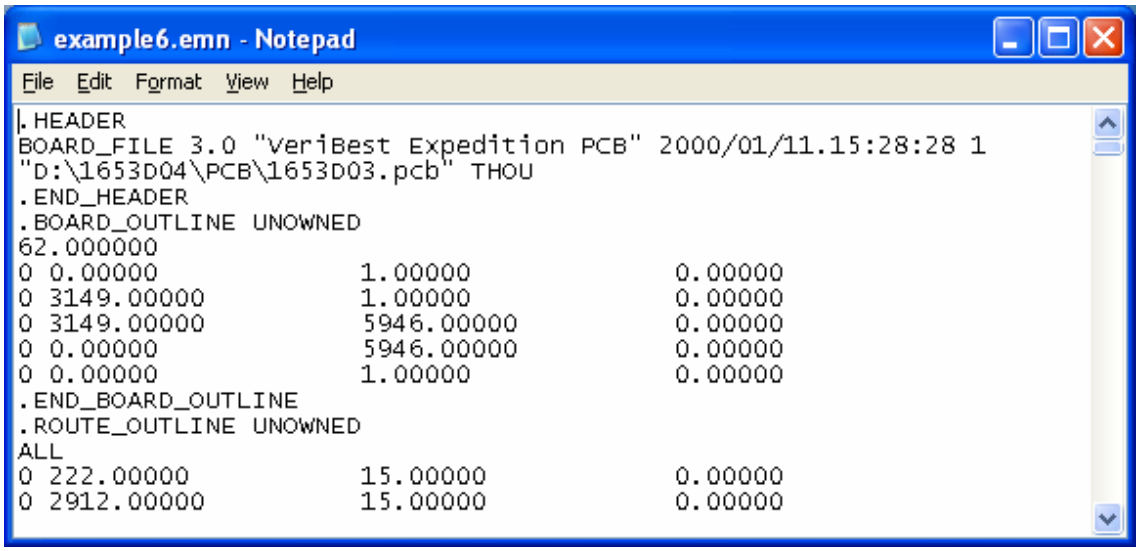
In 1997, responsibility for supporting and enhancing the IDF specification was taken over by a company called Intermedius. More information about the history of the IDF specification can be found on their website at www.intermedius.com

Although commonly referred to as IDF files, each IDF 'file' actually consists of two files; one containing information about the PCB and the components on it, the other containing information about the size and shape of those components. For CircuitWorks to work correctly, both files must be present, and saved in the same location with the same name.

E.g., *'C:\Program Files\CircuitWorks\Example1.enm'* and
'C:\Program Files\CircuitWorks\Example1.emp'

To add to the confusion, there are no standard file suffixes for IDF files. The only thing they never seem to end with is .idf!

The mostly found common file suffix pairs are .emn/.emp, and .brd/.lib, but others are also used. The easiest way to see if the files you wish to use are proper IDF files is to open them with a text editor such as Windows Notepad. A proper IDF file will start something like this:



Most sections in an IDF 2.0 or IDF 3.0 file start with this '.HEADER' format. If in doubt, see if CircuitWorks will open it. If CircuitWorks displays an error message, the chances are you don't have an IDF file.

The Intermedius website (www.intermedius.com) also contains a list of the most common ECAD systems that can write and/or read the IDF format.

Installing and Licensing CircuitWorks

Before you install - System Requirements

CircuitWorks is an add-in to SolidWorks rather than a standalone product, so each seat of CircuitWorks must be installed on a computer with SolidWorks already installed on it. If SolidWorks is not installed, please install it first before attempting to install CircuitWorks.

SolidWorks 2003 or later is required for CircuitWorks 7 to run, however we recommend using the latest version of SolidWorks available, and ensuring it is running the latest service pack CircuitWorks 7 was developed for optimum performance with SolidWorks 2005 and later.

NOTE: To find out which version of SolidWorks you are using, start SolidWorks; select 'Help' from the menus, and then 'About SolidWorks…'. If you are a SolidWorks subscription customer you can download the latest version of SolidWorks from their website at www.solidworks.com

CircuitWorks system requirements are the same as those of SolidWorks. Any computer capable of running SolidWorks adequately will also be able to run SolidWorks. Some IDF files can contain many thousands of components, so the higher the specification of PC the better CircuitWorks will run. For optimum performance we recommend using CircuitWorks on a PC running Microsoft Windows XP®, but CircuitWorks will also work with Windows 2000 and Windows NT. We do not recommend running CircuitWorks on Windows 98 or ME.

Before you install - Downloading CircuitWorks from the Internet

CircuitWorks is normally downloaded over the internet, but may also be provided on a CD. In each case the installation process is similar. If you've received CircuitWorks on a CD, please skip this section and move straight to 'Installing CircuitWorks'

If you have purchased the software from your SolidWorks Reseller (VAR) or directly from Priware Limited, you will be sent an email explaining how to download the software. This will contain a link you can click to start the download process. The link will prompt you to download a file called '*cwf7sp1.exe'* or similar (don't worry if the filename isn't quite the same, it will depend on the current version of CircuitWorks). Save the downloaded file to a temporary location on your computer (if you intend to install CircuitWorks on more than one computer we recommend you download file to a shared drive to save time later).

NOTE: If you've followed the instructions on the website to download an evaluation version of CircuitWorks from the internet, you should have also downloaded a similar file to your computer. In both cases the installation process is the same.

Installing CircuitWorks

Before you install CircuitWorks, please shut down all other programs on your computer that may be running, including SolidWorks. It is also recommended that you suspend the operation of any virus checking software as this may disrupt the installation process (remember to switch it back on once the software has installed). Please also ensure you have sufficient privileges to install software on your PC if you are in a managed network environment.

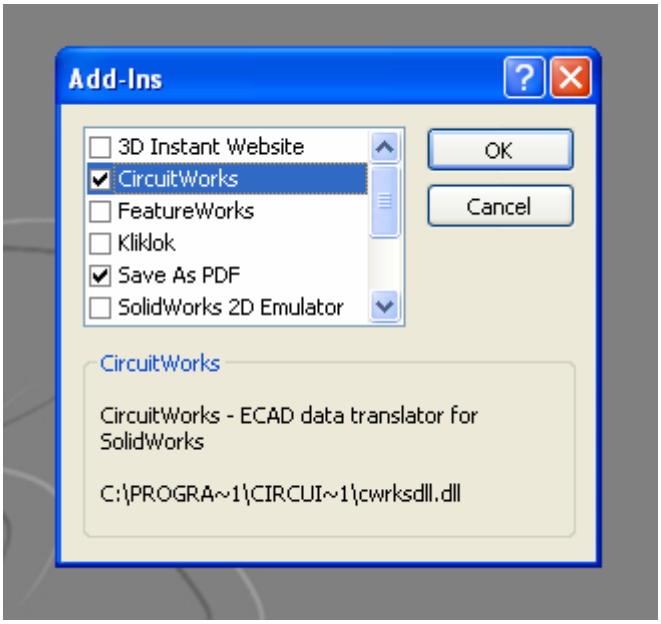
To start CircuitWorks installing, double click the downloaded file in Windows Explorer and follow the on screen instructions. If you have CircuitWorks on a CD, just insert the CD into the CD-Rom drive on your PC and the installation process should start automatically.

If you encounter any problems installing the software, please see the troubleshooting section at the back of this user guide.

Adding CircuitWorks into SolidWorks

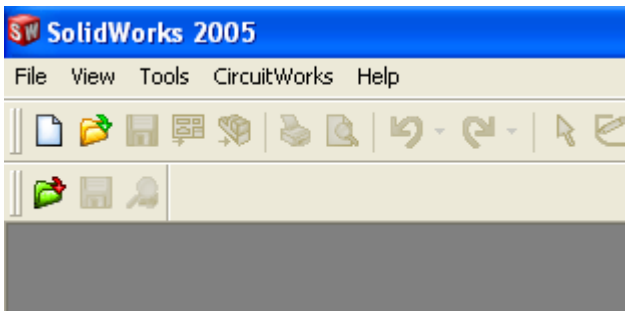
Once CircuitWorks has been installed on your computer, the next step is to ensure it is correctly added into SolidWorks.

Start SolidWorks, and then select 'Add-Ins…' from the 'Tools' menu. SolidWorks will display a list of the currently available add-ins, with a checkbox indicating which are currently added in to SolidWorks. Scroll down the list until you find 'CircuitWorks' and ensure the checkbox is ticked.



Licensing CircuitWorks

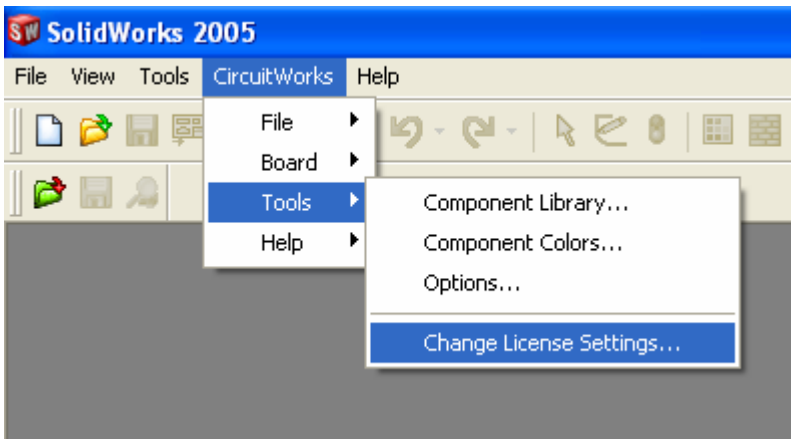
If CircuitWorks is installed and added in to SolidWorks correctly there should now be a 'CircuitWorks' menu in SolidWorks and an additional CircuitWorks toolbar with three icons:



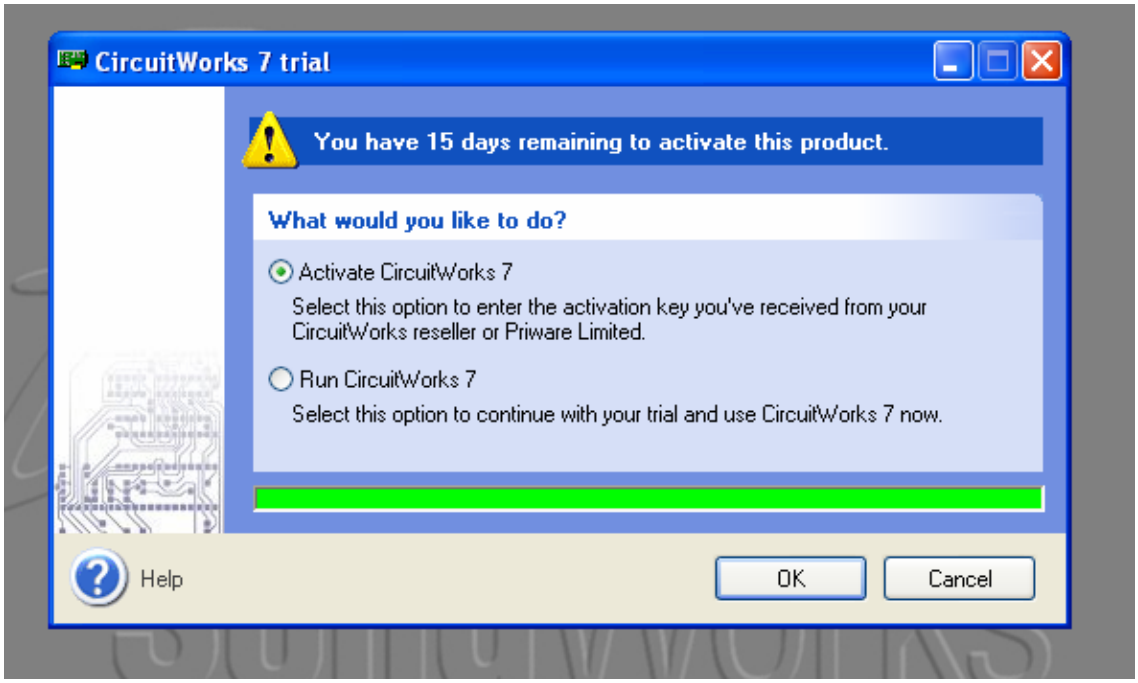
An unlicensed or evaluation version of CircuitWorks will run for 15 days. This allows purchasers of CircuitWorks to begin to use the software even if they have yet to receive an activation code, and also allows people to evaluate the software before purchasing it.

Whilst running in unlicensed mode, CircuitWorks will prompt the user to enter an activation code every time it is first used during a SolidWorks session.

If you have a received a license code or network license file from Priware or your reseller, you can license CircuitWorks by selecting 'Change License Settings…' from the 'Tools' menu under the 'CircuitWorks' menu:



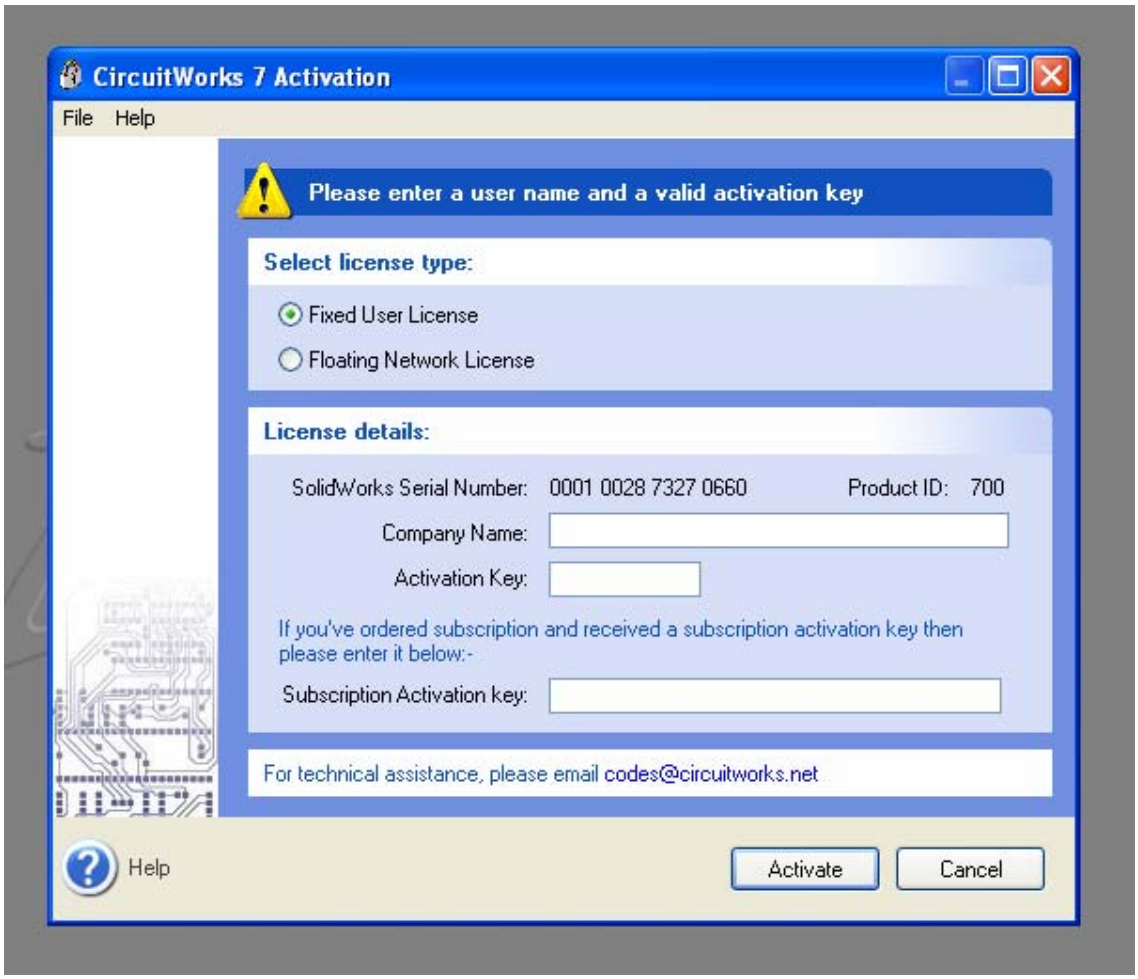
Selecting this menu item will bring up the CircuitWorks license manager dialogue:



This dialogue is displayed every time CircuitWorks is used during the 15 day evaluation period. If you have not yet received a code, or wish to continue to evaluate CircuitWorks then select the second option ‘Run CircuitWorks 7’. You will then be able to continue to use CircuitWorks.

If you have received license details for your copy(s) of CircuitWorks, then accept the default choice ‘Activate CircuitWorks 7’ and click ‘OK’.

CircuitWorks will now display the following dialogue:



CircuitWorks is available in two versions, fixed and floating license. The process for licensing CircuitWorks is determined by the type of license you have purchased:

Fixed License:

Fixed License versions of CircuitWorks are licensed to a fixed seat of SolidWorks. CircuitWorks is activated by a license key that is an encryption of a particular SolidWorks serial number. The seat of CircuitWorks will only operate with a SolidWorks installation with that specific serial number, for that reason care should be taken to ensure you provide your reseller with the correct SolidWorks serial number when purchasing CircuitWorks. Fixed licenses of CircuitWorks cannot be used with floating licenses of SolidWorks.

The SolidWorks serial number is a 16 digit number in the format XXXX XXXX XXXX XXXX which can be found by selecting '*About SolidWorks…'* from the '*Help'* menu in SolidWorks.

Floating License:

Floating License versions of CircuitWorks allow one or more users to share a number of CircuitWorks licenses across several computers. A company with ten SolidWorks users for example could have two floating licenses of CircuitWorks. CircuitWorks could be installed on all ten SolidWorks seats, but only a maximum of two SolidWorks users could use CircuitWorks at the same time.

Floating licenses of CircuitWorks can be used with either Fixed or Floating licensed versions of SolidWorks.

Licensing a fixed license copy of CircuitWorks

Ensure the 'Fixed User License' button is selected in the top part of the dialogue, and check the SolidWorks serial number displayed in lower part is correct for the license you have ordered.

You should have received an email containing your company name and activation code from Priware or your reseller. Carefully type or copy and paste the company name and code into the boxes on the dialogue.

IMPORTANT: For CircuitWorks to work correctly, both the company name and activation code must be correct and exactly as typed in the email from Priware or your reseller. Substituting a different company name format from that in the email, e.g. 'ACME Limited' rather than 'ACME Ltd' will not enable the CircuitWorks license.

If you have purchased subscription for your seat of CircuitWorks, you should have been provided with a subscription code. Please enter this in the 'Subscription Activation Key' box.

Once you have entered your company name, activation code and optional subscription key, click 'Activate' to license your seat of CircuitWorks.

If CircuitWorks displays an error, please double check that the license information you have entered is exactly the same as the information you have been provided with and the SolidWorks serial number is correct. If this still doesn't solve the problem, consult the troubleshooting information at the back of this user guide.

Licensing a floating license copy of CircuitWorks

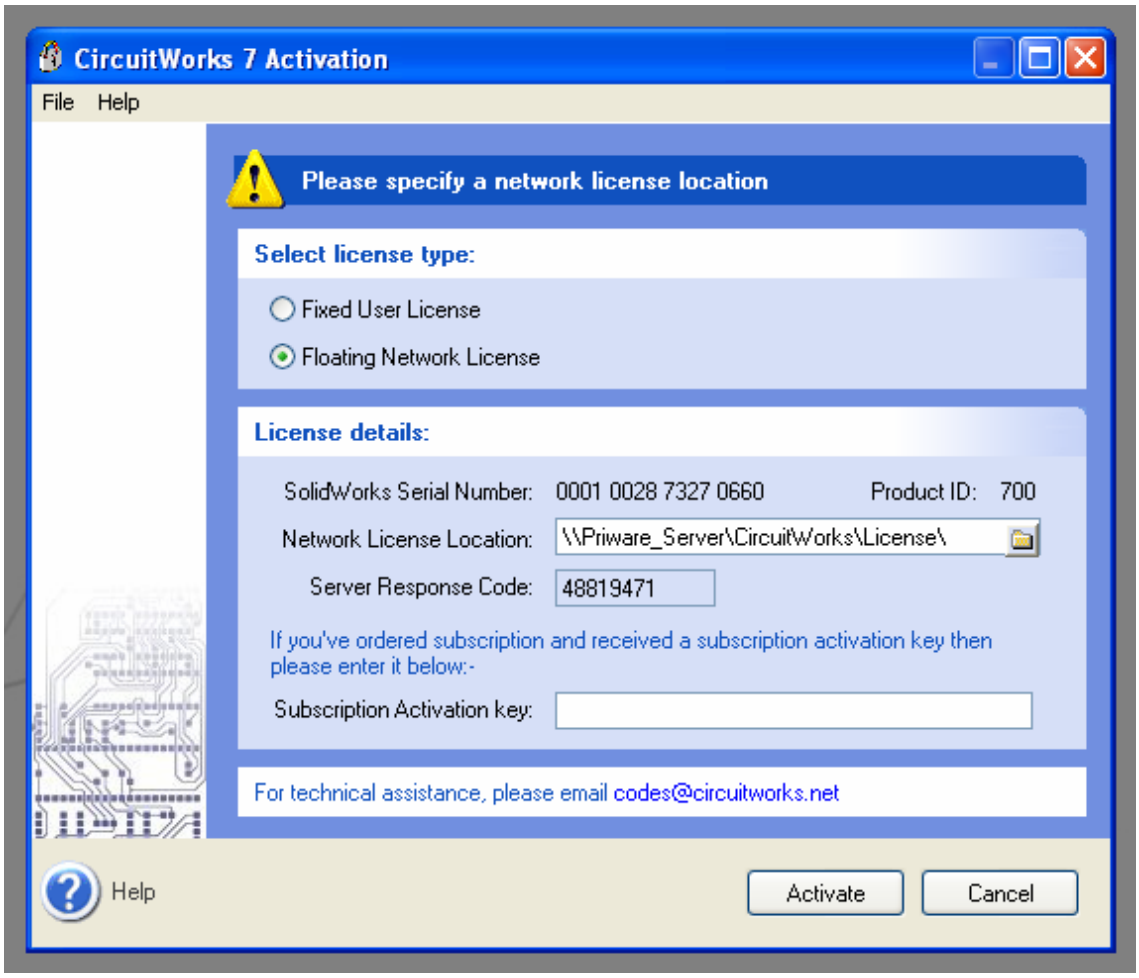
If you have purchased a floating license of CircuitWorks, your reseller will email you a CircuitWorks network license file. This is a small file called '*circuitworks.net'* which contains licensing information.

The CircuitWorks.net license file must be copied to a safe network location that can be accessed by every user who intends to use CircuitWorks, for example //Mech_Server/Licenses/CircuitWorks/ The license file is accessed whenever any seat of CircuitWorks is run, so the file needs to be saved in a secure yet accessible location.

Once the file has been copied to a suitable location, CircuitWorks needs to be set to use the license file. First install CircuitWorks on every PC with SolidWorks that requires it. The copies of CircuitWorks should be running in 15 day evaluation mode, and every time a CircuitWorks

command is accessed, they should prompt the user either to activate CircuitWorks, or to continue with the evaluation period.

To force this dialogue to appear, select *'CircuitWorks / Tools / Change License Settings…'* from the *CircuitWorks* menu in SolidWorks. Then select the default 'Activate CircuitWorks' option and click 'OK'. The following dialogue should be displayed:



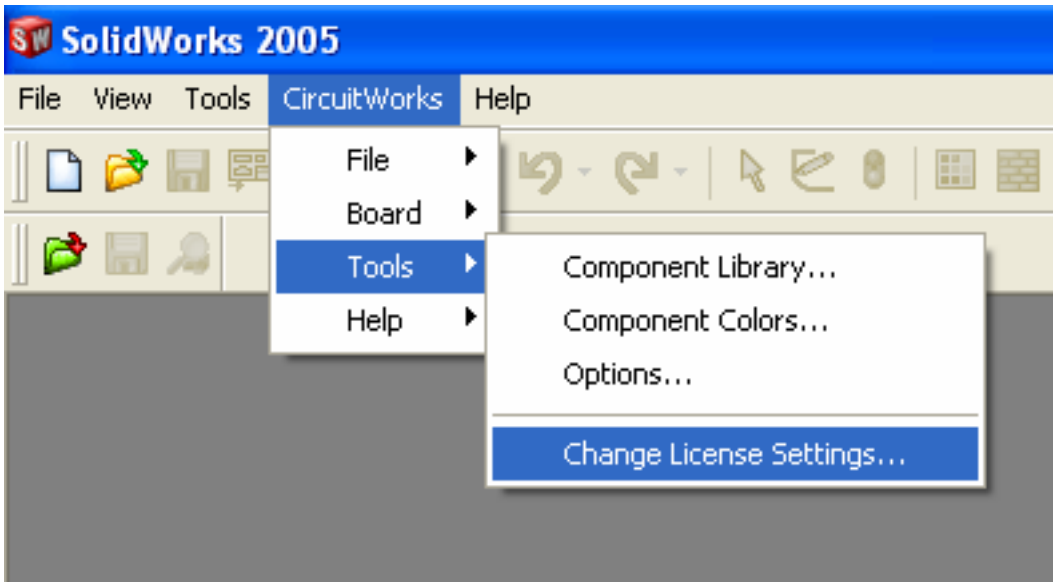
Select the 'Floating Network License' option as shown. The lower part of the form will change to display the License details for the Network License. In the 'Network License Location' text box, enter the UNC path of the location you have saved the 'circuitworks.net' network license in, or browse to it using the button on the right hand side of the text box.

If you have purchased subscription for CircuitWorks, enter your Subscription Activation Key in the box provided as well, and click 'Activate' to license CircuitWorks.

This process must be repeated on every CircuitWorks seat installed on the network license.

CircuitWorks – Overview

CircuitWorks adds an additional toolbar with three icons and a menu into SolidWorks:



All functionality in CircuitWorks can be accessed from the menu, but the icons provide a shortcut to the most commonly used functions in CircuitWorks.

The three icons are (from left to right):

 **Open** – Opens an existing IDF or PADS file and builds a SolidWorks assembly of it.

 **Component Information** – Displays information about the assembly currently open in SolidWorks.

 **Save** – Save the current SolidWorks part or assembly as an IDF file.

The menu structure is divided into four sections; '*File'*, '*Board'*, '*Tools'*, and '*Help'*. The precise functions of the individual menu items are described in the next few sections of the manual, but broadly, their functions are as follows:

File Menu – Commands relating to opening or closing an IDF file
Board Menu – Commands relating to an open board assembly in SolidWorks
Tools Menu – Commands relating to the CircuitWorks application, such as system options
Help Menu – Help and Information

The menus and icons are context sensitive so they will be grayed out if they cannot be used in a certain situation – attempting to find Assembly Information with no assembly open for example.

Processing an IDF file with CircuitWorks

NOTE: Although the term 'IDF file' is used throughout this manual, the method for processing PADS files is identical to IDF files. Some functionality is not available when processing PADS files. For more information please refer to the Appendix at the back of this user guide.

Start SolidWorks, and select either '*CircuitWorks / File / Open IDF File…'* or click *the 'Open IDF File'* icon in the toolbar:

Open IDF File icon: 

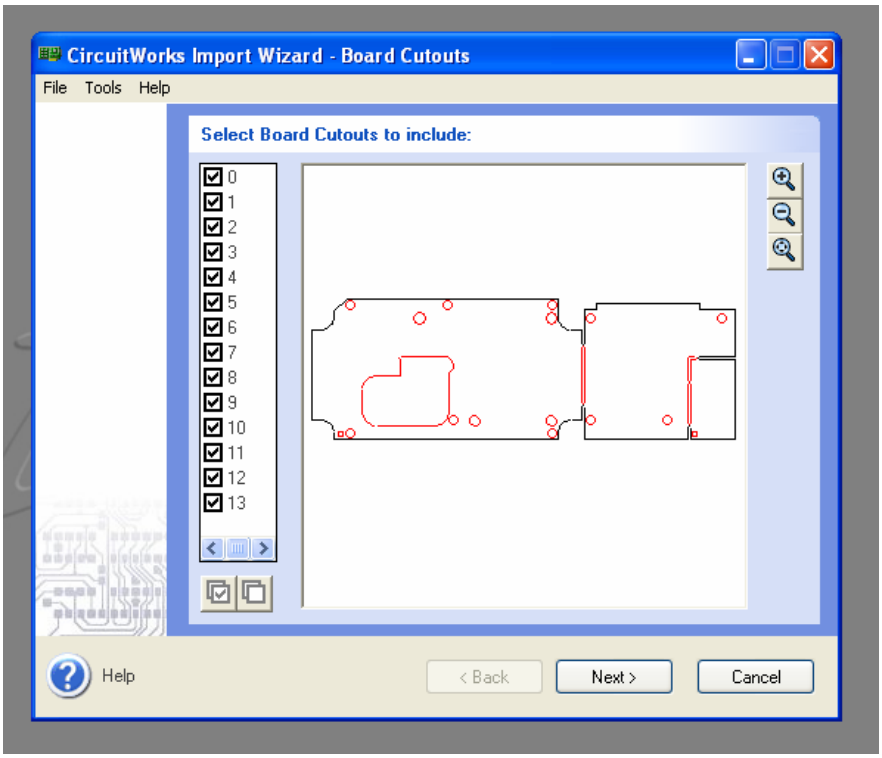
CircuitWorks will now prompt for a file location for a IDF or PADS file. If you don't have any files of the correct type on your computer, select one of the CircuitWorks example files installed in */Program Files/CircuitWorks/Examples*. On first use, CircuitWorks will default to this location.

*NOTE: Each IDF 'file' is actually two files. One contains information about the PCB outline and any holes and cutouts and the other contains the information about the size and shape of the components on the board. The naming conventions of these files vary according to the system producing them, but *.emn & *.emp is the most common. For CircuitWorks to operate correctly, both files need to have the same name and be saved in the same location; e.g., 'example1.emn' and 'example1.emp'.*

Select a file from the list and click open. CircuitWorks will now step through the Import Wizard to assist with the processing of the file. Some of the stages of the Wizard may not be shown if they are not relevant to the IDF file being processed.

Board Cutouts

If the board contains cutouts, the first stage of the Import Wizard will be to determine which cutouts to include in the assembly CircuitWorks will create.



CircuitWorks will show a preview image of the board outline in the window on the right hand side of the page of the Wizard (shown here in black), and any cut outs (shown here in red).

The preview image can be moved by holding down the right hand mouse button in the graphics area and dragging the image. The icons on the right hand side zoom in and out on the image and zoom to extents.

Zoom In

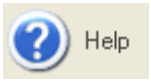
Zoom Out

Zoom to Extents

The checkboxes to the left of the preview image correspond to the number of cut outs found in the PCB – 13 in the case of this example. Un-checking a checkbox will remove the corresponding cut out from the preview image, and hence in the resulting PCB model that CircuitWorks creates in SolidWorks.

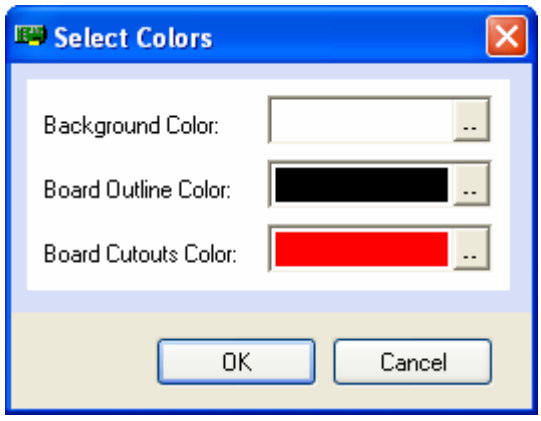
The two icons at the bottom of the list of cut outs allow the user to quickly show or hide all the cut outs in the PCB.

Show all cut outs (left) and Hide all cut outs (right)



To get help on this or at any point in CircuitWorks, click the help icon in the lower left hand corner of the CircuitWorks Window, select *Help / Contents…* from the menu, or hit F1 on the keyboard.

To change the default colours used for the display of the board out line and cut outs, select Tools / Colors… from the menu in the Import Wizard (not the CircuitWorks menu in SolidWorks). The following dialog will be displayed:



Click the buttons next to each colour to change the display colour used in the preview image. To save your changes and exit, click ‘OK’. To close the dialogue without saving your changes click ‘Cancel’.

Non Plated Holes

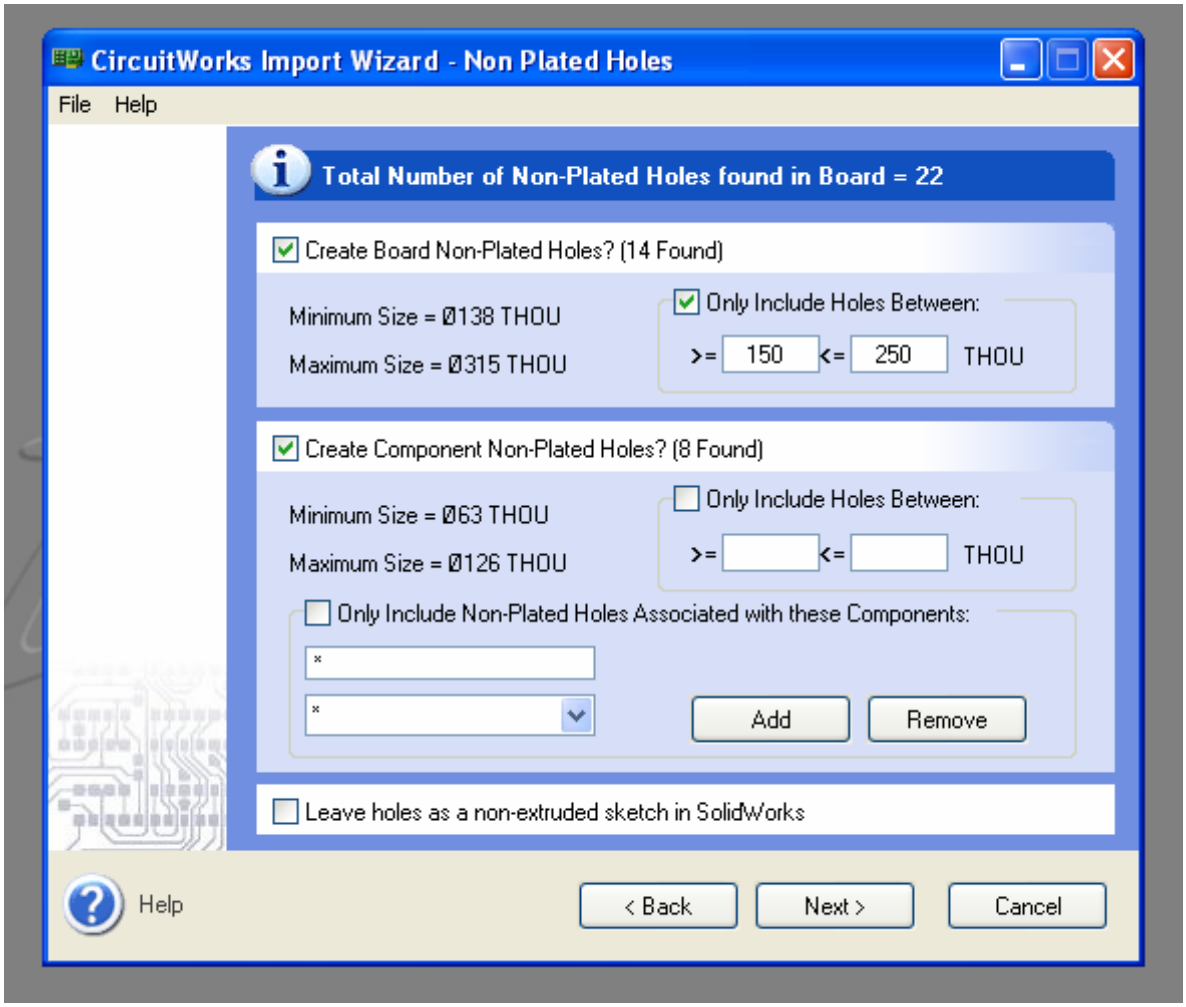
Clicking the 'Next >' button on the Wizard will display the Non Plated Holes dialogue. Non plated holes are holes that have been defined in the IDF file as free from conductive plating. These are usually 'Mechanical' holes, e.g. Holes is the board that are used to hold the board to its enclosure. As a result, these holes are normally the most common type you might wish to include in the SolidWorks assembly.

TIP: Click the '< Back' button at any time to step back to the previous page of the Wizard

There are two types of Non Plated Holes that can be defined in an IDF file; Board Non Plated Holes (holes associated with the board itself), and Component Non Plated Holes (those associated with components).

NOTE: Hole naming conventions vary between ECAD systems, so we recommend you experiment with CircuitWorks and your ECAD system to ensure you get the holes you want in the completed SolidWorks assembly.

CircuitWorks summarises the quantity of Non Plated Holes found in the information bar at the top of the Wizard – 22 in the case of the example below:



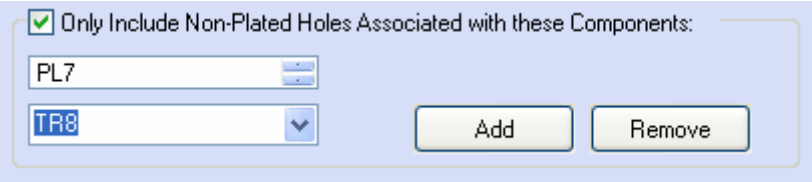
Of the 22 holes found in total in this example, CircuitWorks has found 14 Board Non Plated Holes and 8 Component Non Plated Holes.

Ticking the checkboxes will instruct CircuitWorks to include the holes in the SolidWorks Assembly. In this case, both checkboxes have been ticked, so CircuitWorks will include both the Board and Component Non Plated Holes in the model.

For each hole type, CircuitWorks displays the minimum and maximum sizes found in the IDF file. The user then has the option to filter out holes from the resulting assembly according to their size by checking the 'Only Include Holes Between' option. In this case, only Board Non Plated Holes between 150 and 250 Thou in diameter are to be included in the board.

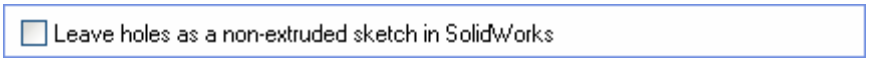
NOTE: Although the units displayed in this example are in Thou, the display units can be set in CircuitWorks / Tools / Options for metric users.

Component plated holes can also be filtered according to their associated component. If this functionality is supported by your ECAD system:



The lower list box contains a list of available components, select one and click 'Add' to add it to the upper list box of components' holes to include. The text box supports wildcards, such as ''*' to represent one or more characters.

Finally, Non Plated holes can be extruded through the board profile in SolidWorks, or left simply as a sketch in the resulting model. By default, holes are always extruded through the board, but if you'd rather just leave the holes as a sketch, select the final option 'Leave holes as non-extruded sketch in SolidWorks'

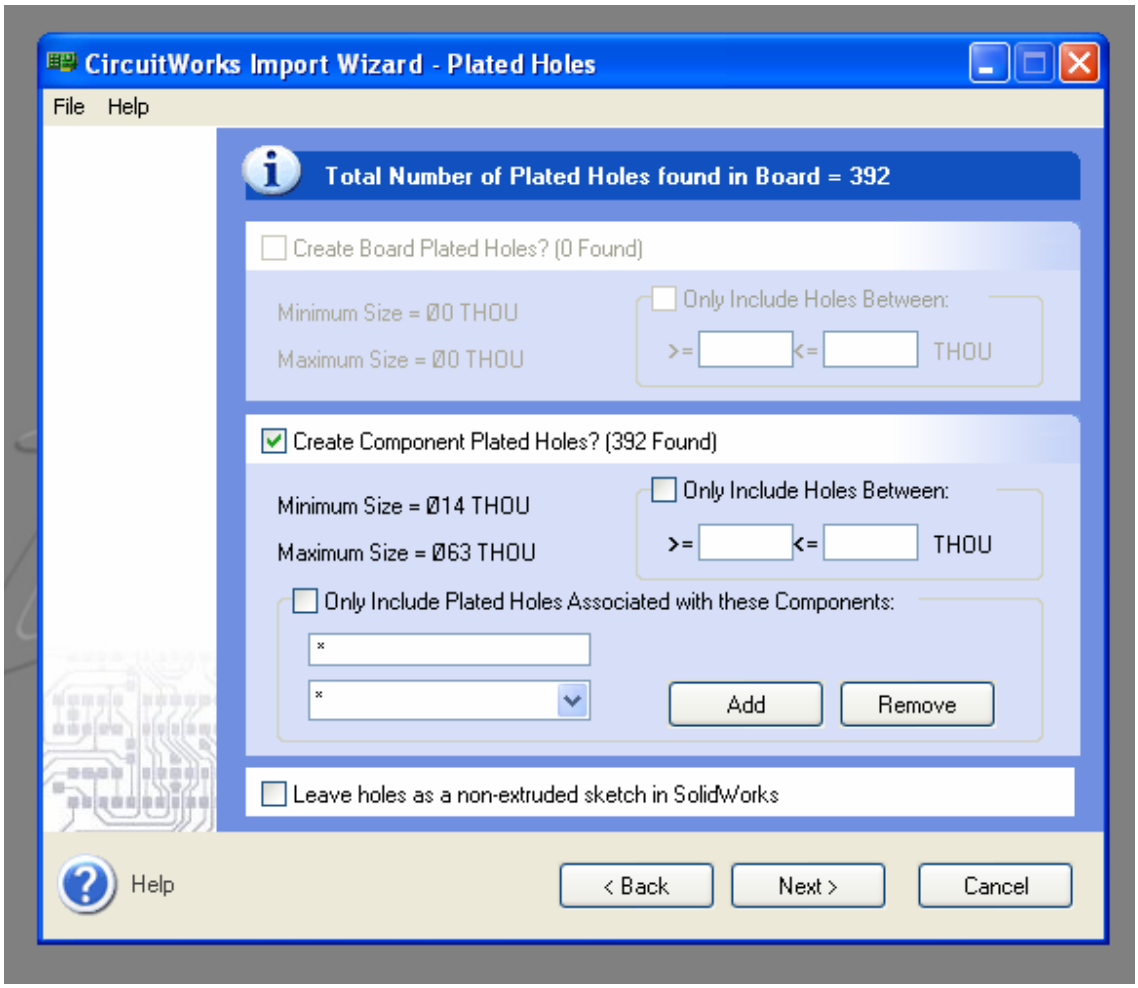


Extruding the holes though the board will produce a more ascetically pleasing result in SolidWorks, but may slow SolidWorks down where many holes are involved.

Once the Non Plated hole options have been set as required, click '*Next* >' to continue to the next page of the CircuitWorks Import Wizard.

Plated Holes

The Plated Hole dialogue is exactly the same layout as the preceding Non Plated Hole dialogue, but controls the creation of holes defined in the IDF file as plated with conductive material. These are generally holes that are associated with the pins of the various components on the board.



In this case, no Board Plated Holes have been found in the IDF file, just 392 Component Plated Holes which have been selected for creation.

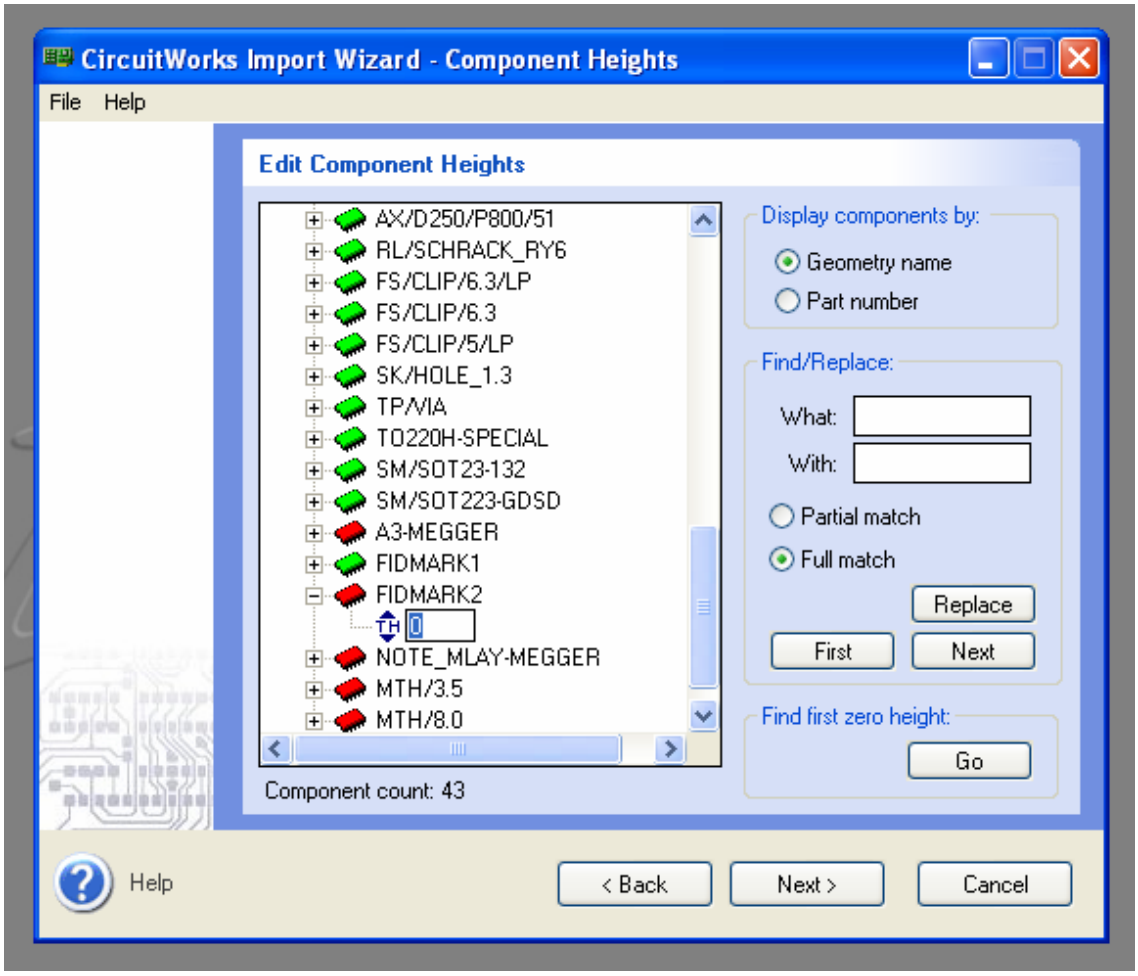
TIP: IDF files tend to contain more component plated holes than any other type, so care should be taken to avoid inadvertently modelling huge numbers of holes in SolidWorks, especially if they are not required in the final model. The speed implications of modelling large numbers of component holes will of course vary with the specification of the computer you are running SolidWorks on. We recommend experimenting with your setup to see what works best with your system.

Once all the Plated Hole options have been set as required, click *'Next >'* to continue to the next stage of the Import Wizard.

Component Heights

Most ECAD packages are basically two dimensional systems, so many do not contain information about the height of the components. In order for SolidWorks to accurately produce a three dimensional model of the board and its components, it requires a height for each new component it encounters if one is not already defined in the IDF file.

When CircuitWorks encounters one or more components in the IDF file that does not have a height defined, it will display the Component Heights dialogue:



The list to the left of the dialogue lists the components CircuitWorks has found in the IDF file. They can be displayed either by Geometry Name or Part Number by selecting the appropriate option to right. In the example above the components are shown by Geometry Name.

The colour of the component icon indicates if it already has a height defined:

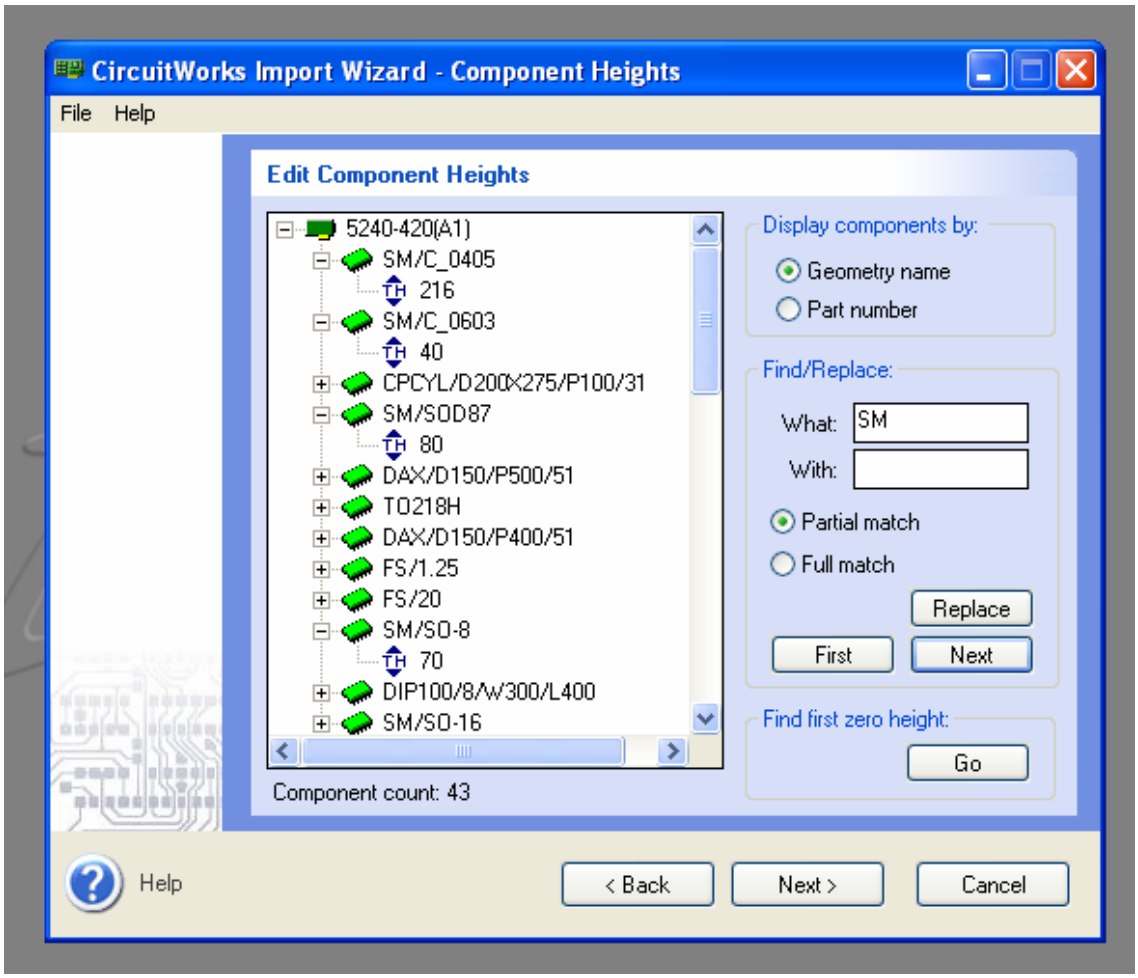
- Height already defined in the IDF file (Height is not 0)
- Height of component is not defined in the IDF file (Height is 0)

To edit the height of a component, click it in the tree to expand the node under it. This will reveal at figure next to a pair of blue up and down arrows. The units between the arrows indicate the units required – Thou in this example.

Enter new figures for the component heights as required. Once a component has a non-zero height, the icon will change from red to green.

In the above example, the height of component ‘FIDMARK2’ is being changed.

To help find a component in an IDF file with a large number of components, ‘Find/Replace’ can be used to quickly sort through the component list for a component or components matching the name specified in the ‘What’ box:



For example, here, ‘SM’ has been entered in the ‘What’ text box, and the ‘Partial Match’ option has been selected.

Clicking the ‘First’ button will jump to the first instance of a component containing ‘SM’ in the component list. Subsequently clicking ‘Next’ will jump to the next instance in the list and so on. Each matching component found will be expanded to show its height as shown in the example above.

‘Replace’ can be used to replace one component name with another.

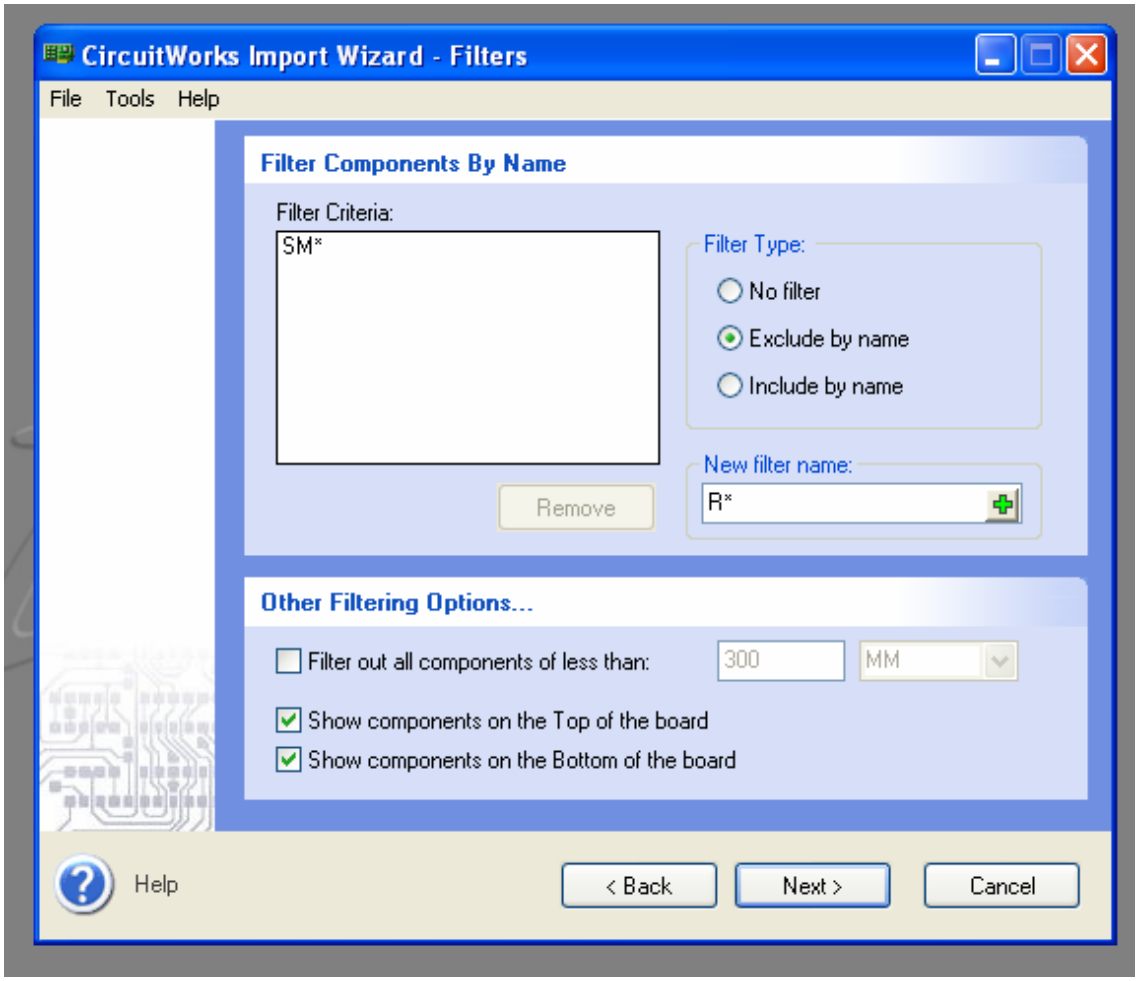
Finally, ‘First 0’ provides a quick way to jump to the first component in the list with a zero height value.

How CircuitWorks treats any components that are left with zero heights is determined by the options in the main CircuitWorks options dialogue in SolidWorks under *CircuitWorks* / *Tools* / *Options…* The options dialogue is dealt with in more detail later in this user guide.

When any component heights have been modified as required, click *‘Next >’* to continue to the next stage of the Wizard.

Filters

Many IDF files contain more component information than is required in the completed SolidWorks assembly. The Filters dialogue allows the user to filter out components from the resulting SolidWorks assembly using various criteria:



The Filter dialogue enables the user to filter out components from the resulting SolidWorks assembly in three ways; by Name, by Height or by Board Side.

Filtering by Name

CircuitWorks can either exclude or include components from the SolidWorks assembly based on their component name.

Say for example we wished to exclude any components whose names begun with ‘SM’ or ‘R’ from our assembly - There might be a large number of surface mount components in the IDF file for example beginning with ‘SM’ which we might not want to show.

To filter out these components, first check ‘Exclude by Name’. This will exclude any component with a name that matches our criteria. If ‘Include by Name’ was checked, CircuitWorks would exclude every component except those that matched out criteria instead.

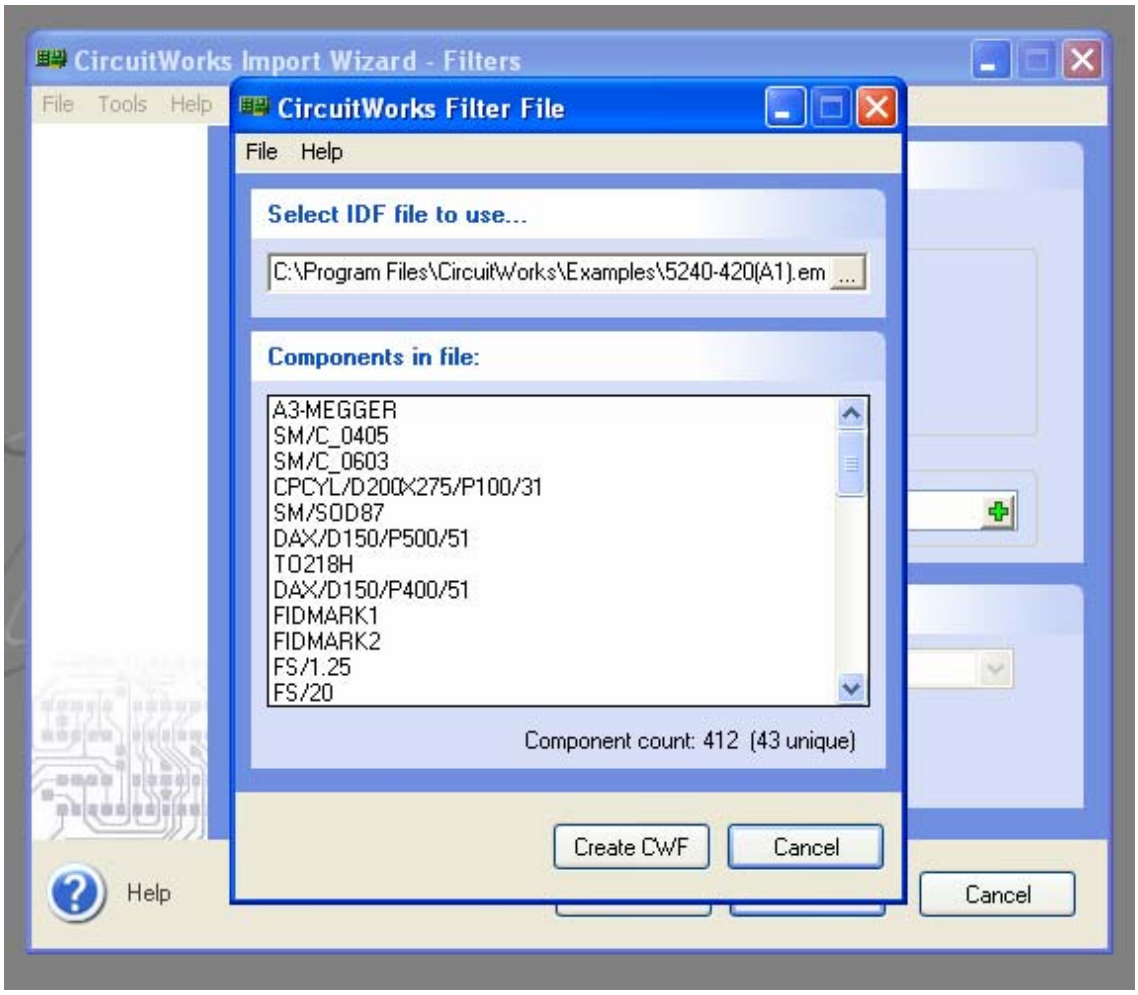
Once ‘Exclude by Name’ is selected, it is possible to type a new filter name in the Filter Name box, so to filter for components starting with SM, type ‘SM*’ and click the green add button to add it to the filter criteria list on the left.

In the screenshot above, ‘SM*’ has been added to the criteria, and ‘R*’ is about to be added. If we were to continue past this dialogue now, CircuitWorks would filter all components beginning with SM or R from the SolidWorks assembly. The ‘*’ as you can see acts as a wildcard (meaning anything), and can be used before are after the entered text.

If there is a particular filter set that you use often – maybe you always want to filter out Surface Mount components for example – it is possible to save the filter settings for easy reuse. Once you have set up your filter criteria, select ‘*File / Save As Filter File…’* from the menu at the top of the dialogue. CircuitWorks filter files are saved with the suffix *.cwf* by default.

To reload previously saved filter settings, simply select ‘*File / Open Filter File’* from the menu and browse to your previously saved filter settings.

As a useful starting point for building more complex filter sets, CircuitWorks can use a list of components from an IDF file to build a filter file which can then be edited as required. To create a filter file directly from an IDF file, select ‘*Tools / Create Filter File from IDF file…’* CircuitWorks will then prompt to browse to an IDF file as shown below:

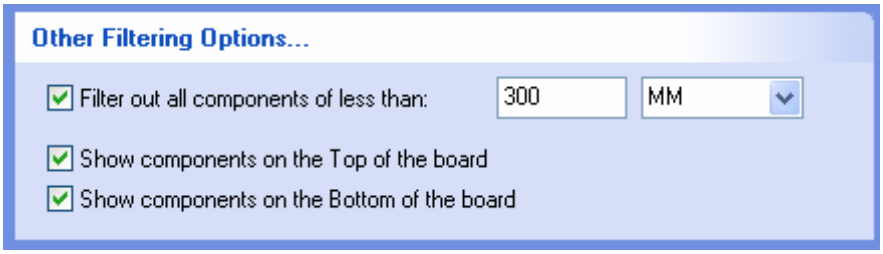


By default, CircuitWorks will list the components in the IDF file you are currently processing, although it is possible to browse to an alternative IDF file to obtain a list of components. Clicking 'Create CWF' will cause CircuitWorks to prompt the user for a name and location for the newly created filter file. The filter file can then be reloaded into CircuitWorks and edited as required.

Note that any filter file created using this method will need to be edited in CircuitWorks to some extent, otherwise, as the file lists every component in the IDF file it will simply filter them all out!

Filtering by Height

Filtering by height is a much more straightforward process than filtering by name. To filter out all components less than a certain height, check the 'Filter out all components of less then' option and enter the height and units. Filtering by height can be useful in situations such as enclosure design, where only taller components would be required in the SolidWorks assembly.



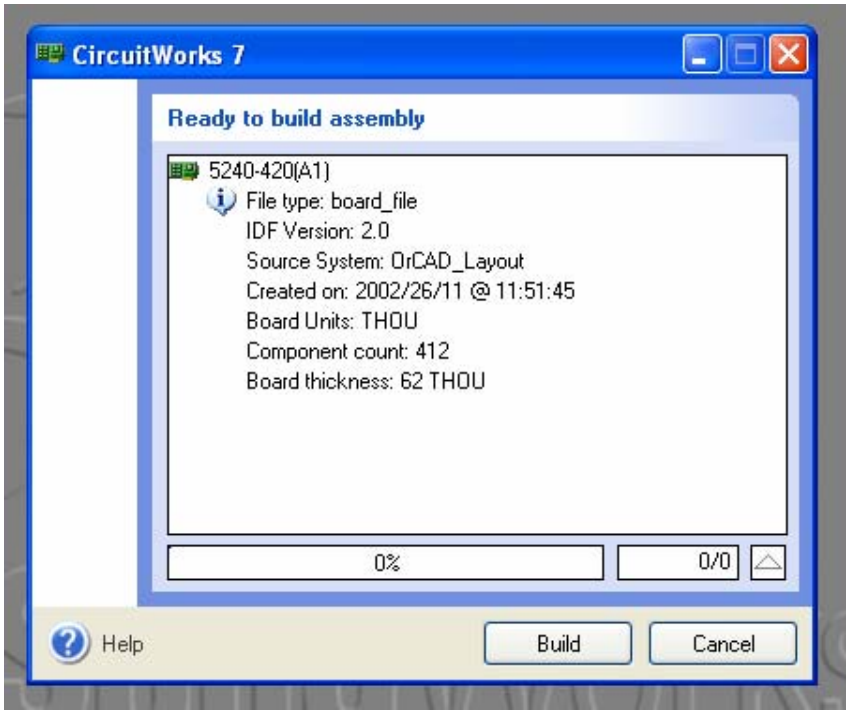
Filtering by Board Side

CircuitWorks fully supports double sided boards and will place components on both sides of the board by default. If only the components on one side of the board are required, uncheck the appropriate box to remove components from the other side. Obviously un-checking both boxes will result in a board with no components on it at all.

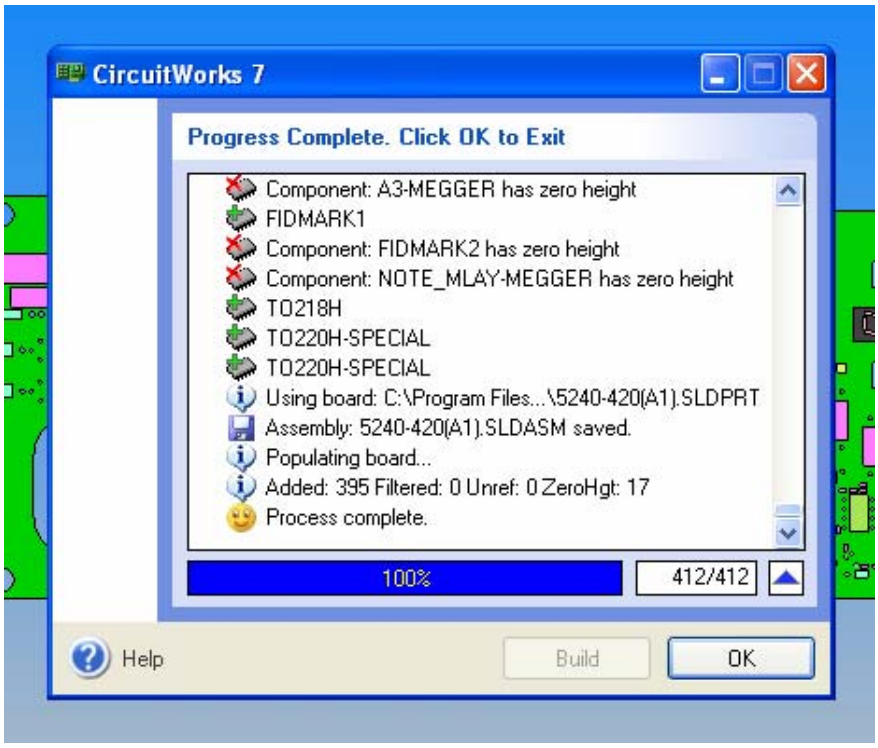
Click '*Next* >' to continue once filtering options are set as required.

Building the Board

Once each stage of the Import Wizard has been completed, CircuitWorks will display a summary of the board information:



The dialogue displays summary information about the IDF file. Click '*Build*' to start processing the file and building the assembly in SolidWorks, or '*Cancel*' to exit.
This dialogue will stay visible to the user whilst CircuitWorks is building the board and components in SolidWorks.



When CircuitWorks has finished processing the IDF file, the Window should look similar to the image at the bottom of the proceeding page.

Assuming the board processed correctly, the progress bar will have reached 100% and the total should display the number of components processed – 412 in this case.

Once CircuitWorks has finished processing the file, CircuitWorks' comments and warnings generated during the IDF file processing can be reviewed if required by scrolling back through the information displayed. The icons beside each comment allow the user to quickly see their meaning using the following key:

 Information

 A component has been successfully added to the assembly.

 Component not added – normally a zero height component or because CircuitWorks cannot find a pre-existing model in the location it was last saved.

 This component has been filtered out by height or name

 CircuitWorks encountered a fatal error and was unable to continue the current conversion.

 Normally indicates an error or warning that you should be made aware of.

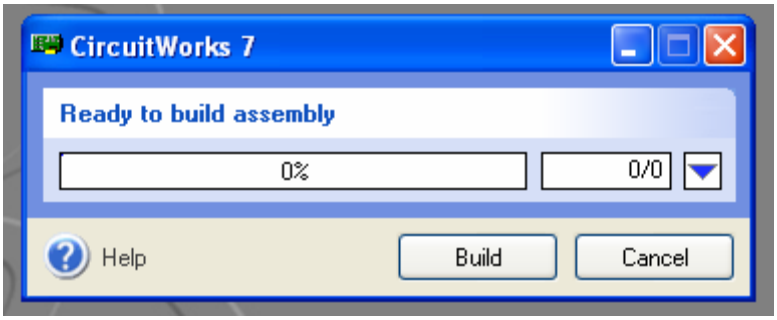
 A component part or assembly is being created.

 A component part or assembly has been saved.

 This indicates that CircuitWorks has changed a parts configuration.

 CircuitWorks has finished processing the current file.

To save space and increase speed, the main CircuitWorks dialogue can be minimized during the build process by clicking the blue arrow as shown:



Click the arrow again to return the window to its original size.

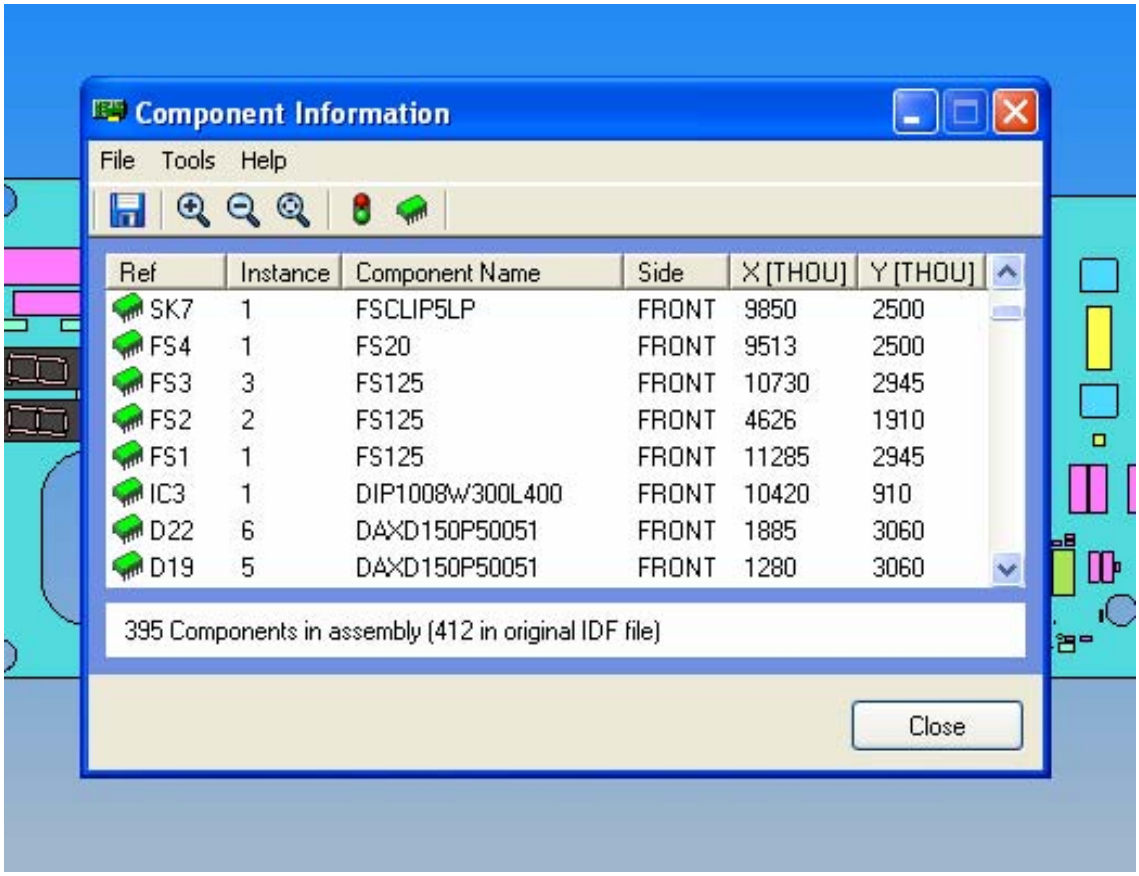
Component Information

Once CircuitWorks has built a SolidWorks assembly CircuitWorks can locate specific components and display information about their names and locations.

With an assembly generated by CircuitWorks open in SolidWorks, select *CircuitWorks / Board / Component Information…*or click the Component Information Icon to display the Component Information Window

 *Component Information Icon*

The component window will appear on top of the current assembly:



The Component Information Window displays a scrollable list of all the components in the current assembly. For each component the following information is displayed:

Ref: The reference designator of the component from the IDF file

Instance: The instance number of the component. In the above illustration for example, there are 3 instances of the component FS125

Component Name: This is the component's name from the IDF file. Note that due to file naming limitations in Windows, the component name will not necessarily be the same as the name of the SolidWorks Part file that represents that component.

Side: The side of the board, Front or Back that the component is placed on

X and Y Values: These are the coordinates of the components' insertion points from the IDF file. To change the units they are displayed in, under *Tools / Set Display Units…*

A summary information bar at the bottom of the window displays the number of components in the Assembly, and the number in the IDF file. In the example shown above, the numbers differ due to zero-height components being filtered out during the build process.

To increase or decrease the width of a column of data, select the edge of the column title and drag it to the left or the right. It is also possible to resize the entire Component Information window, either to display more component information, or to show more of the SolidWorks screen area.

The data can be sorted in either ascending or descending order by clicking a column header. For example, clicking the 'REF' header would sort the components in ascending order by Reference Designator. Clicking the header again will sort the components by descending order by Reference Designator.

Locating a Component in SolidWorks

To locate a component in the list in SolidWorks, click on the component's information row in the component list. SolidWorks will then zoom to the selected component. If the selected component is on the reverse of the board, the board model will be flipped over in SolidWorks, and then the appropriate component will be zoomed in on.

To zoom in more on the component, click the 'Zoom In' icon on the Component Information Toolbar, to zoom out from the component, click the 'Zoom Out' icon. To zoom to fit the whole board to the screen, click the 'Zoom Extents' icon.

Component Information Toolbar

The default zoom level can be changed by selecting *Tools / Set Zoom Level* in the Component Information Window menu structure. The level can be set to Low, Medium or High to indicate how close into a component SolidWorks will zoom when one is selected in the list. A tick indicates the current zoom level.

Locating a Component in the Component List

It is also possible to select a component model in the SolidWorks graphics area and highlight its information in the Component Information Window: Select an component in the SolidWorks graphics area, and then click the 'Locate Selected Component' icon.

Locate Selected Component Icon

The component's row will then be highlighted in the Component Information Window with a green bar.

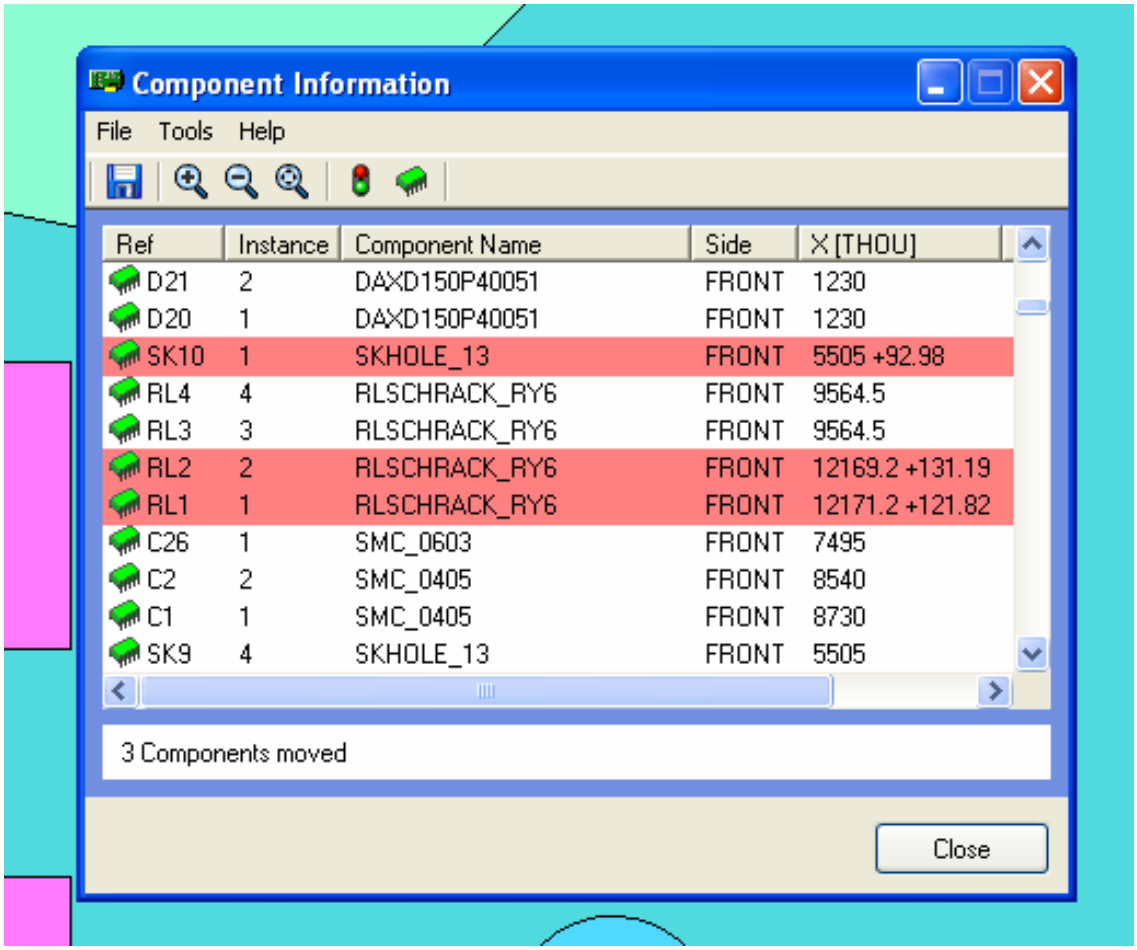
Showing Changes to the Assembly

The Component Information Window can also be used to show any changes that have been made to the SolidWorks assembly since it was created, by comparing the assembly's component positions to those in the original IDF file.

To show changes, move one or more components, and then click the 'Refresh Component List' icon in the Component Information Window:

Refresh Component List Icon

Any components that have been moved will be highlighted with a red band in the Component Information Window:



In this case, 3 components have been moved. Note that the X and Y coordinates update to show the dimensional change that has been made to the component's position.

Saving the Component Data

The data in the list can be saved for future reference or use in another system if required. To save the component data, click the ‘Save’ icon on the toolbar, or select *File / Save Data As…* from the menu structure.

CircuitWorks will prompt the user for a filename and location for the data file. Note that the default location can be set in the main CircuitWorks Options dialogue - see later in this user guide for more information.

CircuitWorks will display a confirmation message if the data has been saved successfully.

The data is saved as a comma delimited text file, which can be read by a large number of systems. Note that this file is just a dump of the data shown in the Component Information Window – not enough information is provided to recreate the board and components. CircuitWorks IDF export should be used to save IDF data from a SolidWorks assembly instead.

Save As IDF

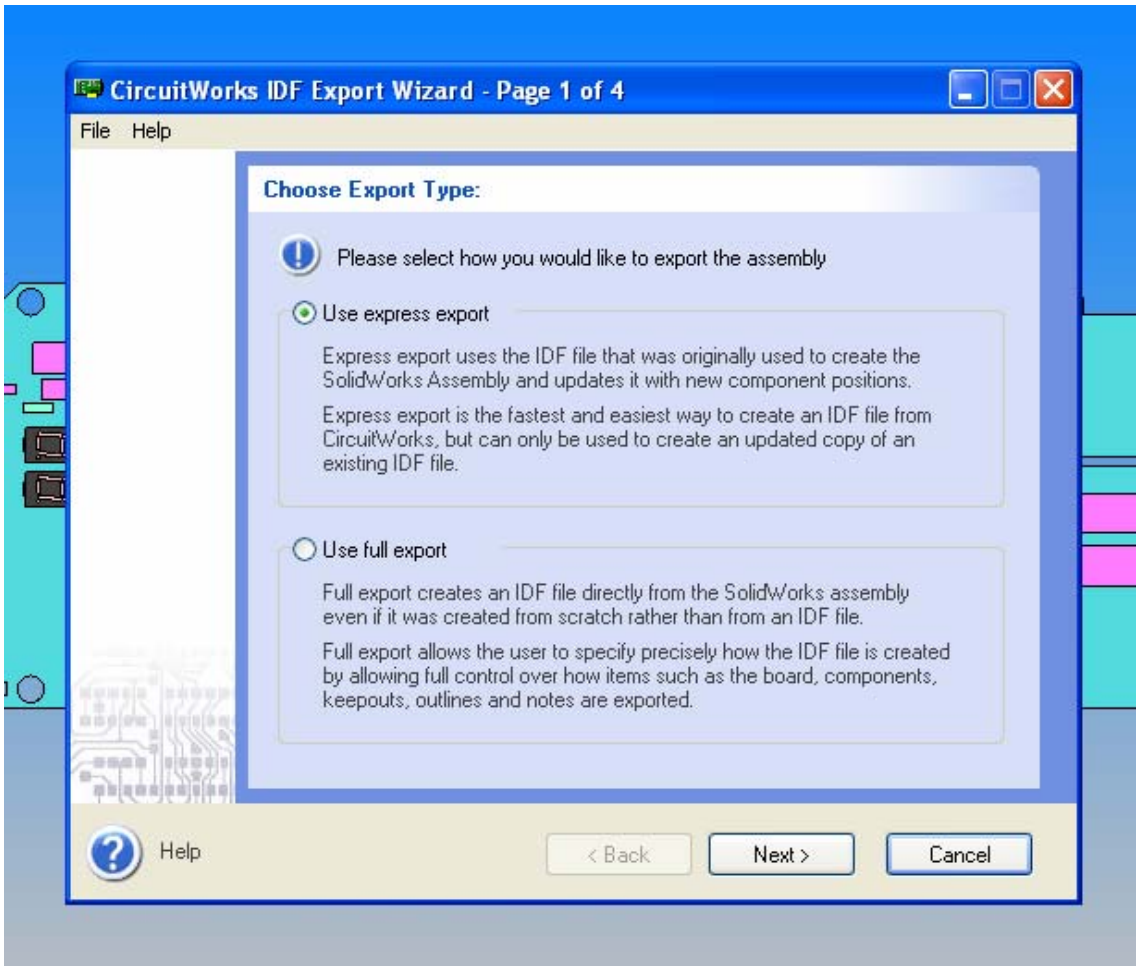
CircuitWorks can save SolidWorks Parts and Assemblies back out as IDF files for use in an ECAD system. This can be useful for reflecting components that have been moved or changed in SolidWorks (perhaps as a result of a clash for example) back into the ECAD system. Changes to the shape of the board, cutout information and holes can also be exported.

CircuitWorks can either generate an IDF file from scratch, or copy and modify an existing IDF file to reflect the changes made. Which approach is used will depend on the amount of changes that have been made to the board and its components, and if the assembly was initially created by CircuitWorks, or the user.

To Save a SolidWorks assembly as an IDF file, select *CircuitWorks / File / Save As IDF…* from the CircuitWorks menu in the SolidWorks menu structure, or click the *Save As IDF* icon in the CircuitWorks Toolbar:

 *Save As IDF*

Clicking the *Save As IDF* icon will display the following dialogue in SolidWorks:



The first page of the Export Wizard offers the user two different methods to create an IDF file from the active SolidWorks Assembly – Express Export or Full Export.

Express Export

Express Export can be used if the assembly you wish to export was originally produced by CircuitWorks, rather than modeled directly in SolidWorks by the user. It works by making a copy of the original IDF file and updating the information within it. The only changes that will be recorded by Express Export are those to component positions. If the assembly you are working on was originally generated by CircuitWorks, and all you have done is change the positions of components on the board, then you should use the Express Export option.

Full Export

Full Export works by analyzing the assembly currently open in SolidWorks and generating an IDF file from the information. It does not require an existing IDF file, or for the assembly to have been generated by CircuitWorks. If the assembly in SolidWorks has been modeled manually, or you have made major changes to an assembly generated by CircuitWorks, such as changing the shape of the board or components, or adding components, then you should use the Full Export option.

The Full Export option can export assemblies that have been modeled from scratch in SolidWorks as well as those that have been created by CircuitWorks. However, if an assembly that has not been initially modeled by CircuitWorks is to be exported as an IDF file, certain conventions need to be adhered to in order to ensure the assembly exports correctly:

The board sketch must be defined on the first plane of the board part and named BOARD_OUTLINE. Any sketch entity found in this sketch will be exported as the board outline, and any internal closed profiles will be exported as board cutouts.

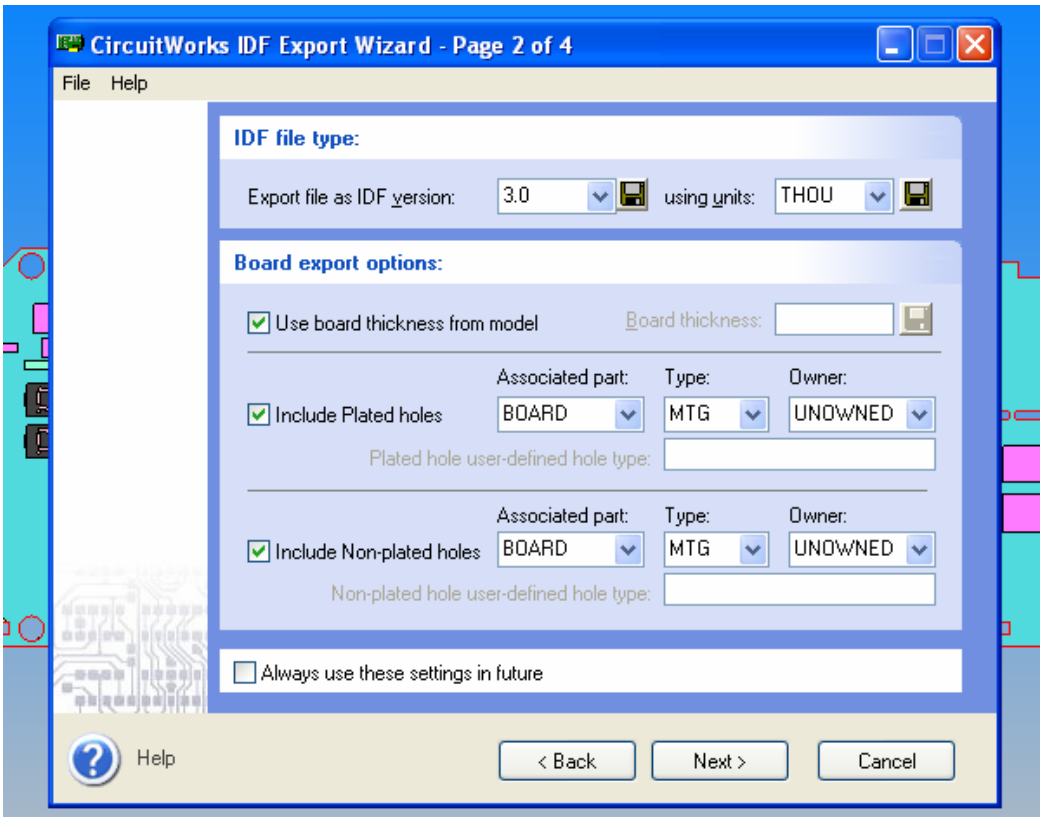
Sketch entities found on sketches called PTH and NPTH will be exported as plated and non-plated drilled holes respectively.

*NOTE: These sketch names can be changed. See the **Export Options** section of this guide.*

Select the option you wish to use to export the current assembly, and click ‘Next >’ to continue to the next stage of the Export Wizard.

If Express Export was selected, CircuitWorks will prompt for a path and filename for the newly created IDF file. By default, CircuitWorks will save the files out with the most common .emp/.emn suffixes used by most ECAD systems. Use the ‘*Save As Type’* pull-down list to select another suffix pair if required, and click ‘*Save’* to save the IDF file.

If Full Export was selected, clicking ‘Next >’ will display the second page of the Export Wizard:



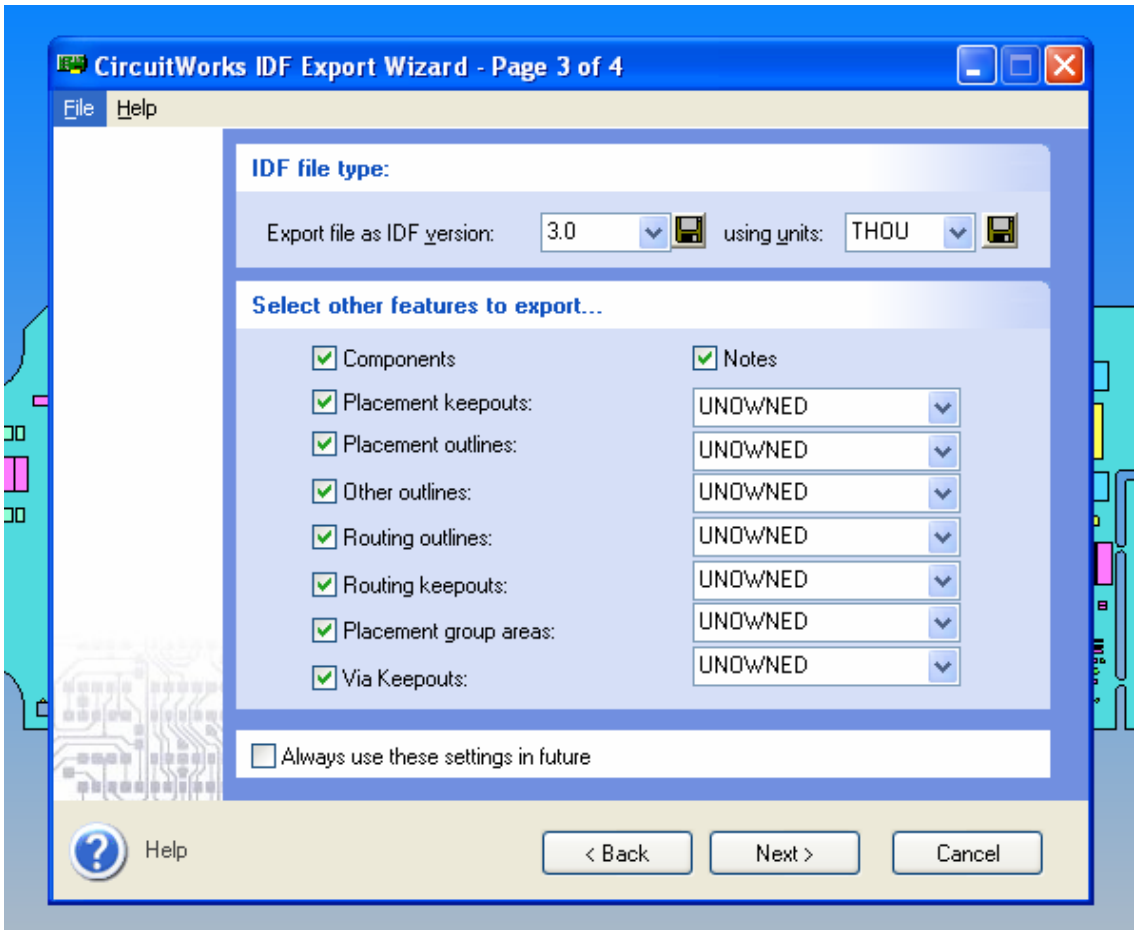
Page 2 of the Export Wizard allows you to chose the format to export the IDF file in – IDF 2.0 or IDF 3.0. IDF 3.0 contains more information than the earlier IDF 2.0 format, but cannot be read by all ECAD systems. Check the documentation of your ECAD system to determine which format to use.

Changing the IDF version disables or enables other settings on the Wizard if they are not supported by that particular file format.

The Export units pull-down determines the units that CircuitWorks should write the file in. This will have no effect on the physical size of the output board and components, but should be set to match the units of the system the IDF file is to be imported into once it is written if possible.

Further options on this page of the Wizard can be used to set the board thickness (calculated automatically from the SolidWorks model by default) and whether to included Plated and Non-Plated holes in the IDF file. If holes are to be included, further options can be set depending on whether the file is to be written in IDF 2.0 or IDF 3.0 format.

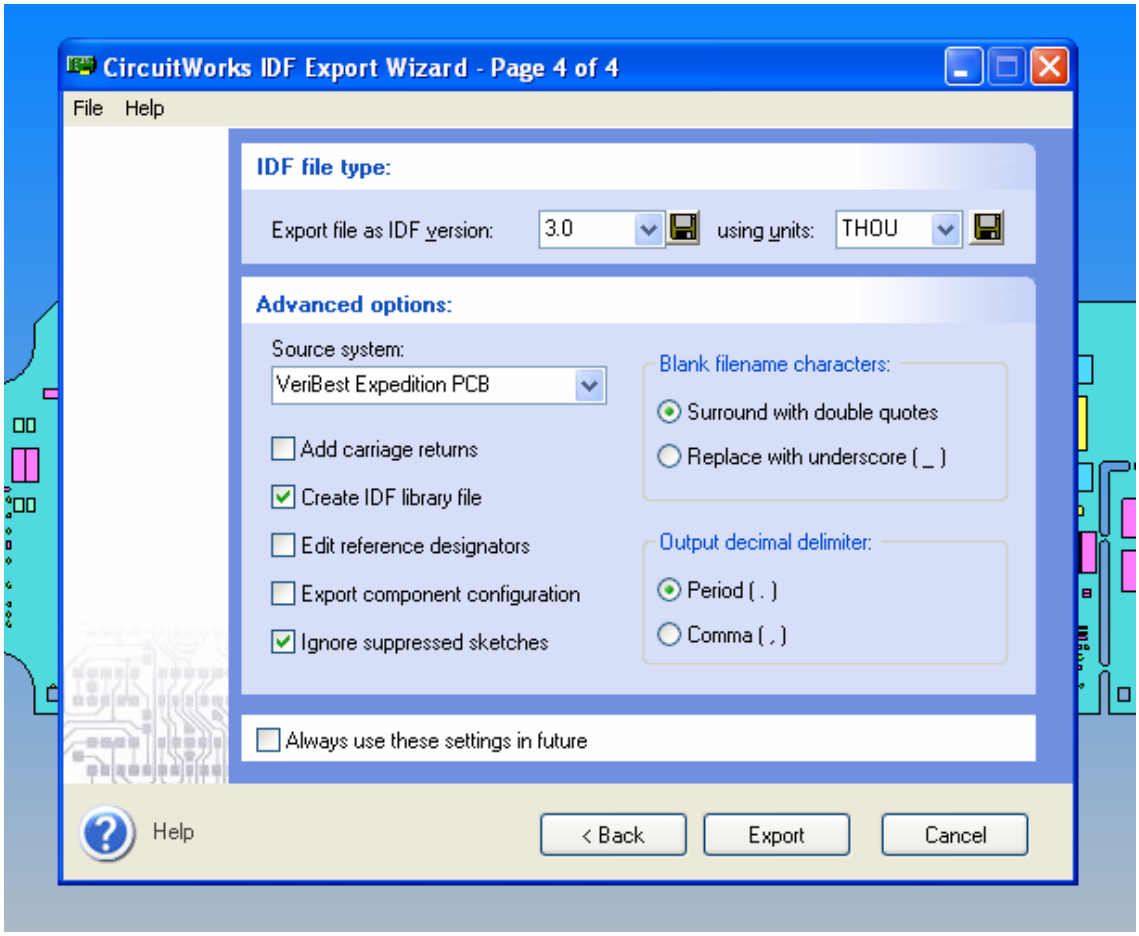
Click the '*Next >*' button to continue to page 3 of the Export Wizard:



Page 3 of the Export Wizard determines which other features should be exported from the SolidWorks assembly. Note that even if all these options are checked, CircuitWorks will only export those features if appropriately names sketches and features can be found within the SolidWorks assembly.

The IDF 3.0 format allows ownership of the profiles to be set via the drop-down lists.

Click '*Next* >' to continue to the final page of the Export Wizard.



Page 4 of the Export Wizard controls more advanced Export options. Setting the source system (in the above example set as Veribest Expedition) identifies the name of the source in the IDF file. Normally, this would be set as 'CircuitWorks Export' as that is the name of the system writing the file. In some cases however, setting the source system to a system other than can be useful. One example is underside component rotation direction (see the *Options* section of this user guide for more information) which can vary according to the system writing the file.

Create IDF library file prompts CircuitWorks to create both parts of the IDF file, and should be switched on by default.

Edit reference designators prompts CircuitWorks to display an additional dialogue to allow the user to change the reference designators for any components in the assembly. If no reference designators are provided, CircuitWorks will allocate them automatically by default.

The other advanced options are provided for 'tweaking' settings that may cause problems for certain ECAD systems. Although IDF is meant to be a standard format, some systems read and write it slightly differently from others, and these settings can help.

If you have specific difficulties reading IDF data exported from CircuitWorks back into your ECAD system then the best approach is to compare a file written by CircuitWorks with one written by your ECAD systems by opening them in a text editor such as Notepad to see if you can spot any obvious differences.

Add carriage returns determines whether to add a carriage return character (ASCII 13) to the end of each line in the IDF file. This is required by some systems.

Export Delimiter determines whether to use a comma (,) or period (.) to delimit records in the file. Again, different IDF systems use different formats, as do systems installed on different International versions of Windows.

Once all settings have been set as required, click '*Export'* to start the export process. CircuitWorks will prompt for a filename for the IDF file to be exported, and will report the export progress back to the user via a window in the Export Wizard Dialogue

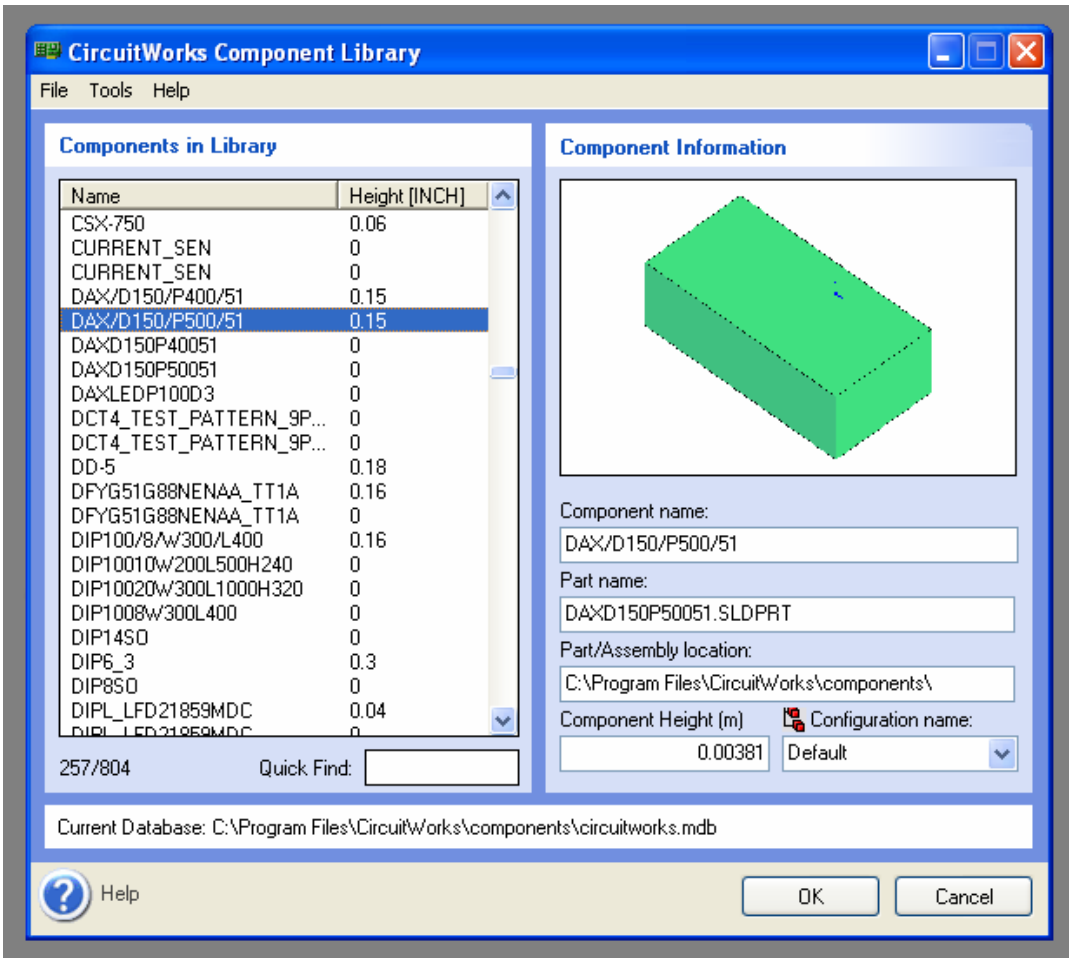
Click 'OK' to close the Export Wizard once the export process is complete.

Component Library

When it is first used, CircuitWorks' component library is empty. As CircuitWorks processes IDF files, it checks its library to see if a component matching the name and height of the component it is processing exists in its internal database. If it exists, CircuitWorks will insert the pre-existing model into the assembly, if not, it will create a new component model using the footprint and height information contained within the IDF file and add it to the library. Consequently, CircuitWorks will become faster the more it is used, as the less time it will spend modeling components.

Ideally, CircuitWorks would save the component models it creates with the same name as the component name in the IDF file, and in most situations this is the case. However, in some cases the component names in IDF files contain characters that cannot be used in Windows filenames such as '*' and '.', so these are stripped out of the filenames of the components CircuitWorks creates.

Once you have run one or more IDF files through CircuitWorks so the library contains some components, open the CircuitWorks library by selecting *CircuitWorks / Tools / Component Library…* from the CircuitWorks menu in SolidWorks. The following dialogue will be displayed:



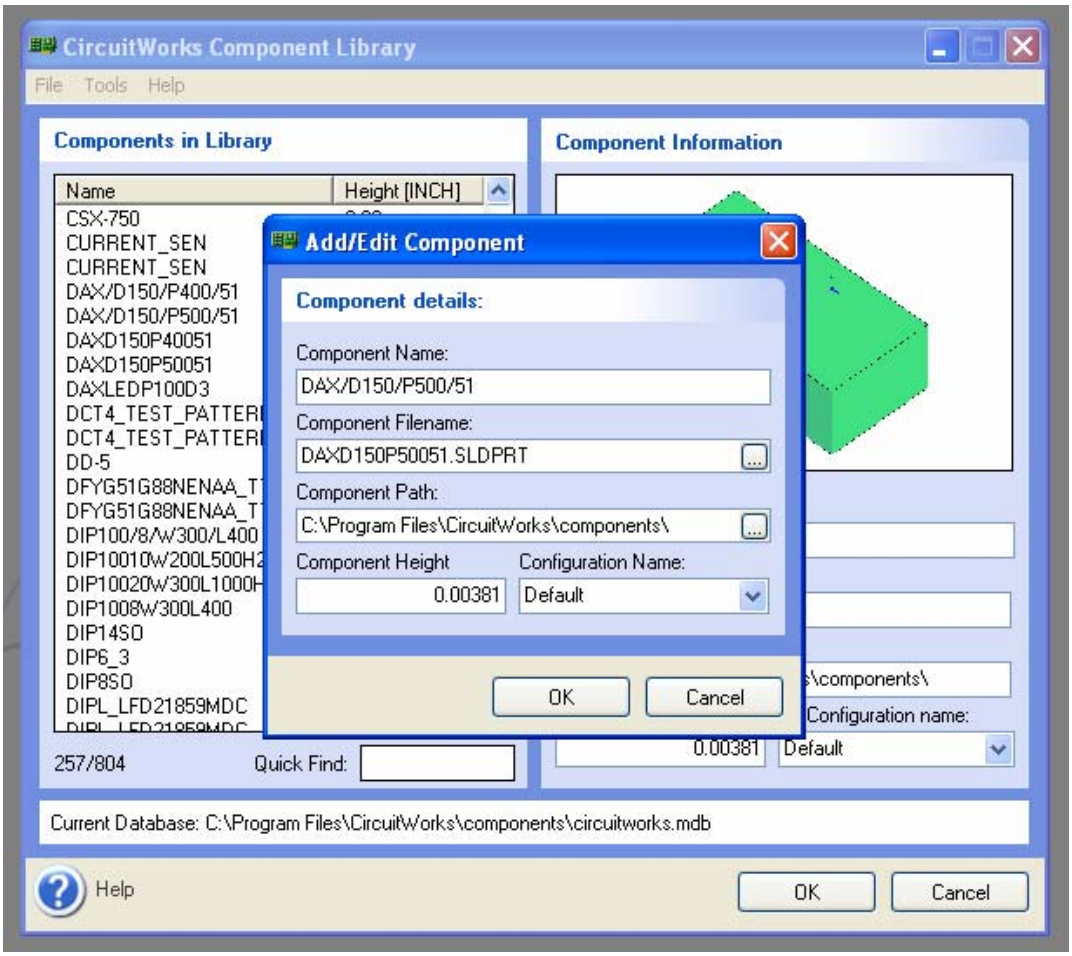
The library lists all the components that CircuitWorks has built for every board it has encountered since it was installed or the library contents were last deleted. The list view on the left hand side of the dialogue lists the name of each component, and its height. CircuitWorks uses a combination of these two things to determine if it has previously encountered a component. The height display units can be changed by selecting *Tools / Set Display Units* in the CircuitWorks Library menu.

For example, in the above screenshot, the selected component is called 'DAX/D150/P500/51' and has a height of 0.15 Inches. If CircuitWorks encountered a component in another IDF file called 'DAX/D150/P500/51' with the same height, it would use the pre-existing model. However, if it encountered a component with the same name but a different height, it would build a new model for that height, so both would be in the library.

Matching components on a combination of name and height, rather than name alone allows CircuitWorks to support ECAD systems where the same component might be drawn in different ways according to its installation – The same resistor lying flat or soldered up on end for example.

Selecting a component in the list on the left will show a preview image of the component and its location information on the right. Typing directly into the 'Quick Find' box will jump the list to the closest match to whatever is typed into the box.

To edit the information for a component, click to highlight it in the list, and select *File / Edit Component…* from the menu in the CircuitWorks Library. The following dialogue will be displayed:



The component name, path, height and configuration can edited as required. To save the changes click ‘OK’, click ‘Cancel’ to close without saving.

CircuitWorks will work fine without any need to edit any components in the library, but some of the reasons for manually changing the library are:

One model for many components – A lot of components might have different names, but be mechanically represented by the same package. For example, one solid model might suffice for a huge range of resistors if each value resistor was physically the same size. In this situation, disk space could be saved by manually setting the path and filename of each resistor in the library to the same single SolidWorks model.

Use of multiple configuration parts – Similar components such as Dual-In-Line (DIL) Packages could be modeled as components of a single SolidWorks part.

Use of assemblies – In some situations you may wish to swap out the component model CircuitWorks has created for a SolidWorks assembly.

Use of user modeled parts – The component models created by CircuitWorks represent the ‘worst case’ of the physical shape of the component, and so are useful for clash detection in SolidWorks. Their simple shape also simplifies the process of using Fluid Dynamics (CFD) programs on the PCB assembly. However, in some situations it would be preferable to show photorealistic component models rather than simple blocks. In this case, the components created by CircuitWorks can be swapped out for more realistic models*

Note that any replacement models or assemblies must be modeled in the same orientation as the models CircuitWorks produces or they will be located incorrectly in their assemblies. The best starting point is to take a model CircuitWorks has produced and modify it to look more realistic. To open a component in the library in SolidWorks to edit it, simple double-click it in the list view or select *File / Open Component* from the menu.

Adding and Deleting Components

To delete a component from the library, click on it in the list view to highlight it, and select *File / Delete Selected Components* from the menu or press the Delete key on your keyboard. Multiple components can be deleted at the same time by holding down the Ctrl key while selecting items from the list view.

When deleting components from the library CircuitWorks can also delete its SolidWorks part file from disk if required.

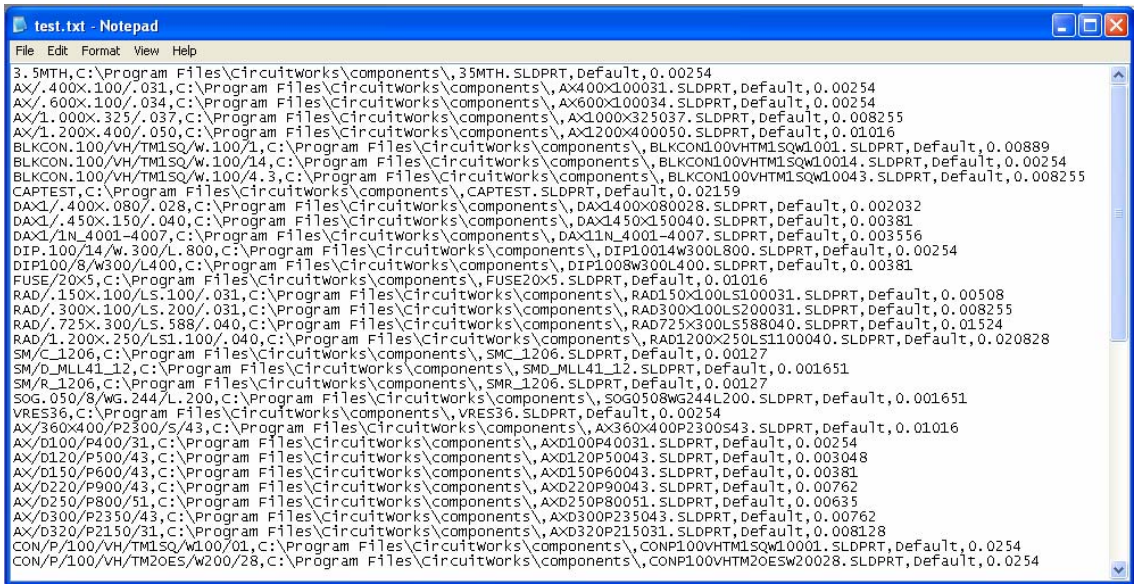
Components can also be added manually to the library by selecting *‘New Component…’* from the *File* menu in the CircuitWorks library and browsing to the required file. As CircuitWorks adds new components to the library automatically, adding components manually is normally not required.

Import Data from File

There a various methods to manually import data into the CircuitWorks component library if required. They are most frequently used when recreating an existing CircuitWorks setup in a new location, where the user does not want CircuitWorks to recreate parts from scratch.

Import Data from File can be found under the *Tools* menu in the CircuitWorks Library. It can be used to import a comma-delimited text file containing component details. Four fields are required on each line each delimited by a comma: the Component Name; its Full Path; the Configuration name to be used, and the Component Height in meters (CircuitWorks stores all heights internally in meters irrespective of how heights are displayed to the user).

A typical data file might look like this:



Opening this file from the ‘Import Data from File’ utility would add these components to the library.

Export Data

Export Data under the tools menu will prompt the user to save out a comma delimited text file of the current library contents in the same format as required by Import Data from File above. The two commands can be used together to interchange library information between two seats of CircuitWorks, or between CircuitWorks and another system.

Import Data from Folder

Import data from Folder can also be found under the *Tools* menu in the CircuitWorks library This option will import the SolidWorks parts from a specified folder into the library. Selecting this option will automatically import all the SolidWorks Parts in that folder into the library.

Note that this option will not physically move the SolidWorks Parts, just add a reference to them in the library. The user will have to manually edit the component heights of the imported components to ensure CircuitWorks matches them rather than creating new models.

Remove Unreferenced Components

Remove Unreferenced Components under the *Tools* menu will remove any component references from the CircuitWorks Library that do not have a corresponding SolidWorks part file on disk. It can be a useful utility for tidying up the library.

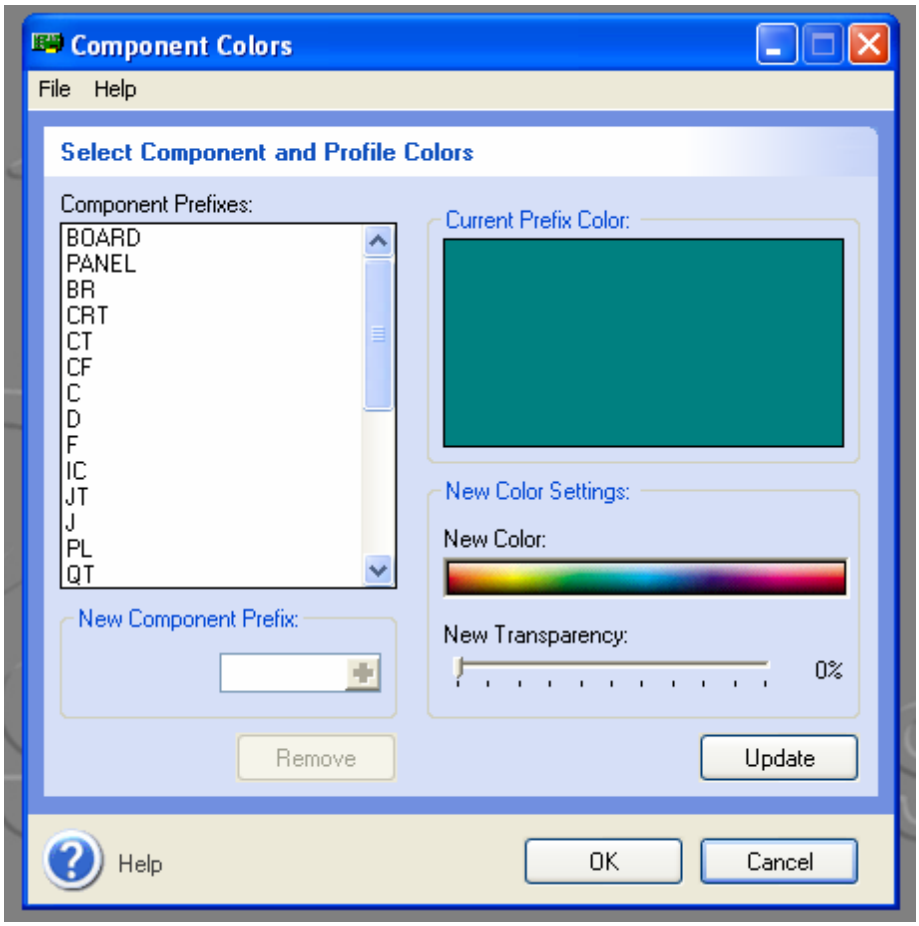
Find and Replace Component Location

CircuitWorks Component Library also contains a Find and Replace utility under the *Tools* menu. It can find text in the component location field of each component in the library and replace it with other text. It can be useful when the library is moved to another location. For example, if every component in the library was created in *'C:\Program Files\CircuitWorks\Library'* and then the components were moved to another drive or location, say *'D:\Program Files\CircuitWorks\Library'*, CircuitWorks internal database would still expect to find the components on the C:\ drive. Finding 'C:\' and replacing it with 'D:\' would update all the paths in the Component Library to reflect the new location.

Component Colors

When CircuitWorks builds a component model, its color is determined by the first letter or letters of its component name. CircuitWorks does this to visually differentiate different types of components to user, in order to make the assembly clearer to the user.

The Component Colors dialogue under *CircuitWorks / Tools / Component Colors…* can be used to change the colors CircuitWorks assigns to new component models:



The list on the left hand side of the dialogue lists the predefined Component Prefixes CircuitWorks is aware of on first installation. Selecting a Prefix from the list will show the color assigned to that prefix in the box in the top right hand side of the window. For example, if ‘C’ is selected in the list, the color shown on the top right will be that assigned to all components with names beginning with ‘C’, e.g., ‘CAP CHIP1024’ or ‘Capacitor 10uF’.

To select a new color for a component prefix, select the required prefix from the list, and use the color bar control to select a new color.

 *Color bar control*

To use the color bar control, hold down the left hand mouse button and drag the mouse cursor over the control. The color will change in the upper color box to indicate the new color.

As well as defining the colors for components, the Component Color dialogue also displays the default colors for the Board itself, as well as for Component Keepout and Outline areas if they are defined within the IDF file.

Note that to display Keepout and Outline areas in CircuitWorks, the appropriate build options must be set – see the Options section in this user guide for more information.

The transparency level of Keepouts and Outlines can also be set using the Transparency Slider control.

Once you are happy with the new color settings for a component, click ‘Update’ to update the default color settings for that component.

Note that any changes made will only affect new components created after the changes have been made. Any components already in the library will retain their existing colors.

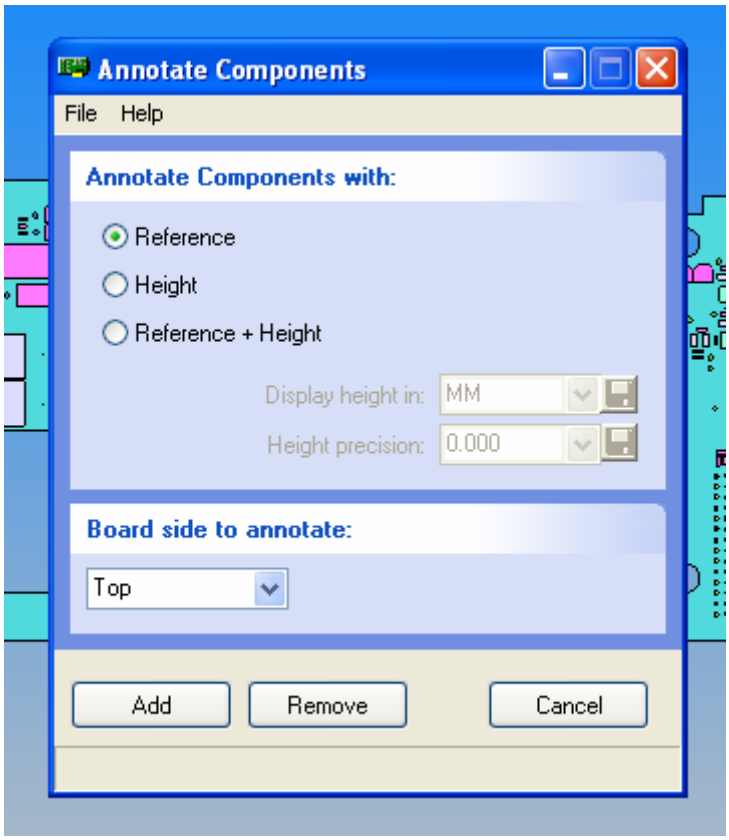
To add a new Component Prefix to the list, type it into the ‘New Component Prefix’ text box and click the green ‘Add’ icon (the green cross). A color can then be associated with it as detailed above.

To remove a component prefix from the list, select it and click the ‘Remove’ button. System defined prefixes such as ‘BOARD’ cannot be removed from the list.

Annotations

Once an assembly has been generated by CircuitWorks, it can be automatically annotated with the components' Reference Designators and/or heights.

To add annotations to a CircuitWorks assembly, select *CircuitWorks / Board / Annotate Components…* from the CircuitWorks menu in SolidWorks to display the following dialogue:

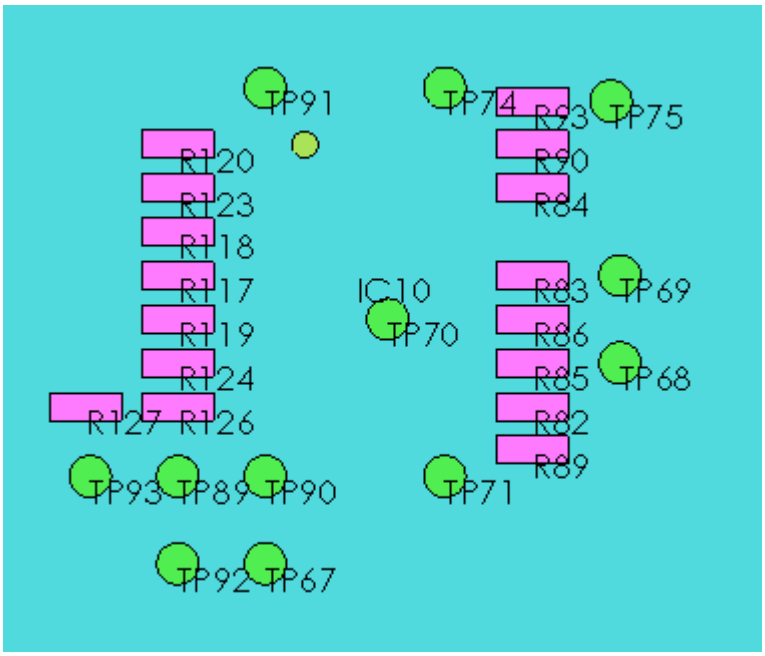


Use the option buttons to choose to annotate the components with either their Reference Designators, Heights, or both.

If Component Height is selected, a choice of display units and precision is available.

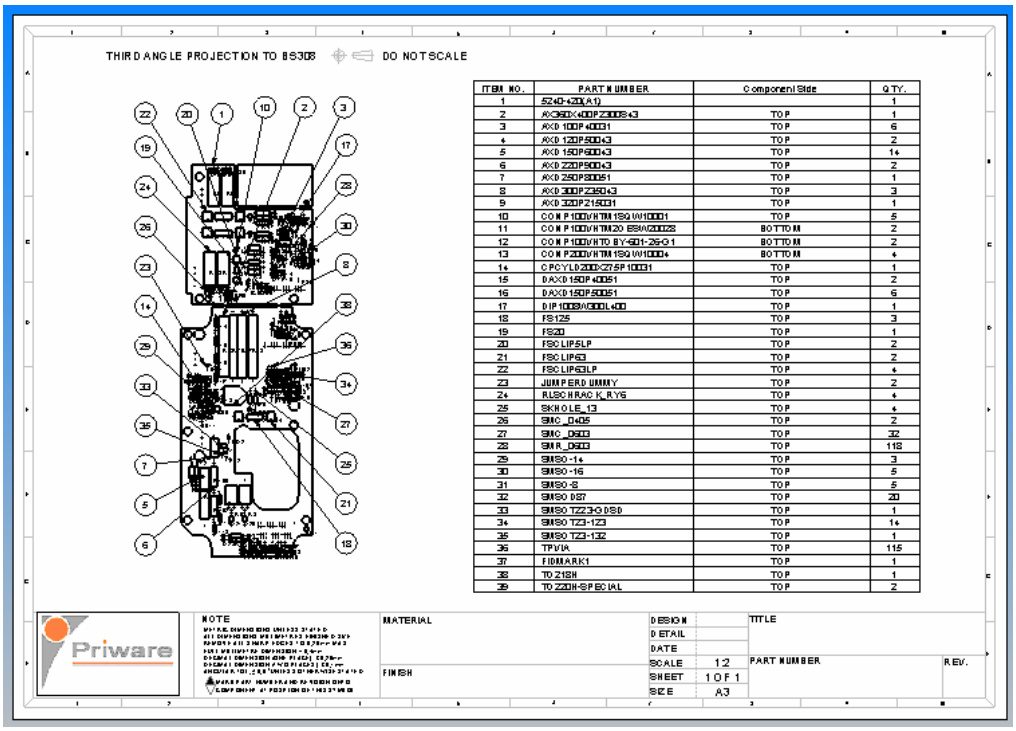
Either the Top, Bottom or Both sides of the board can be annotated – select which side(s) are required, and click 'Add' to add the annotations. To remove existing annotations click 'Remove'.

Once the board is annotated it will appear like this: (detail shown)



Annotations can be displayed on SolidWorks assembly drawings. The precise method of doing this depends on the version of SolidWorks. Please see the SolidWorks Help for more information.

CircuitWorks also writes Custom Property information into the component part files it creates. These custom property values can be mapped to the SolidWorks BOM in an assembly drawing as shown below:



CircuitWorks saves the following SolidWorks custom property values inside each component part file it creates:

Geometry_name (text) – The component's geometry name from the IDF file
Part_number (text) – The component's part number from the IDF file
Component_side (text) – The side of the board the part is on (TOP or BOTTOM)
Component_placement (text) – The component's placement (PLACED or UNPLACED)
Component_height (number) – The component's height in meters.

These are the default names for the custom properties – they can be changed to match existing BOM templates or PDM system requirements in the CircuitWorks Options Dialogue.

Other Utilities

Other useful assembly utilities are grouped together under the *CircuitWorks / Board* menu in SolidWorks including, Fix/Float components, Show Reference Designators and Show Component Names.

Fix Components and Float Components

By default, component models are inserted into the correct location in a SolidWorks assembly, but left unconstrained to float. This saves a huge amount of the time it takes CircuitWorks to create an assembly, but isn't ideal for all assemblies.

If this is the case, the components in the assembly can be fixed in place by selecting *'Fix Components'* from the *Board* menu. To float the components again, select *'Float Components'* from the menu.

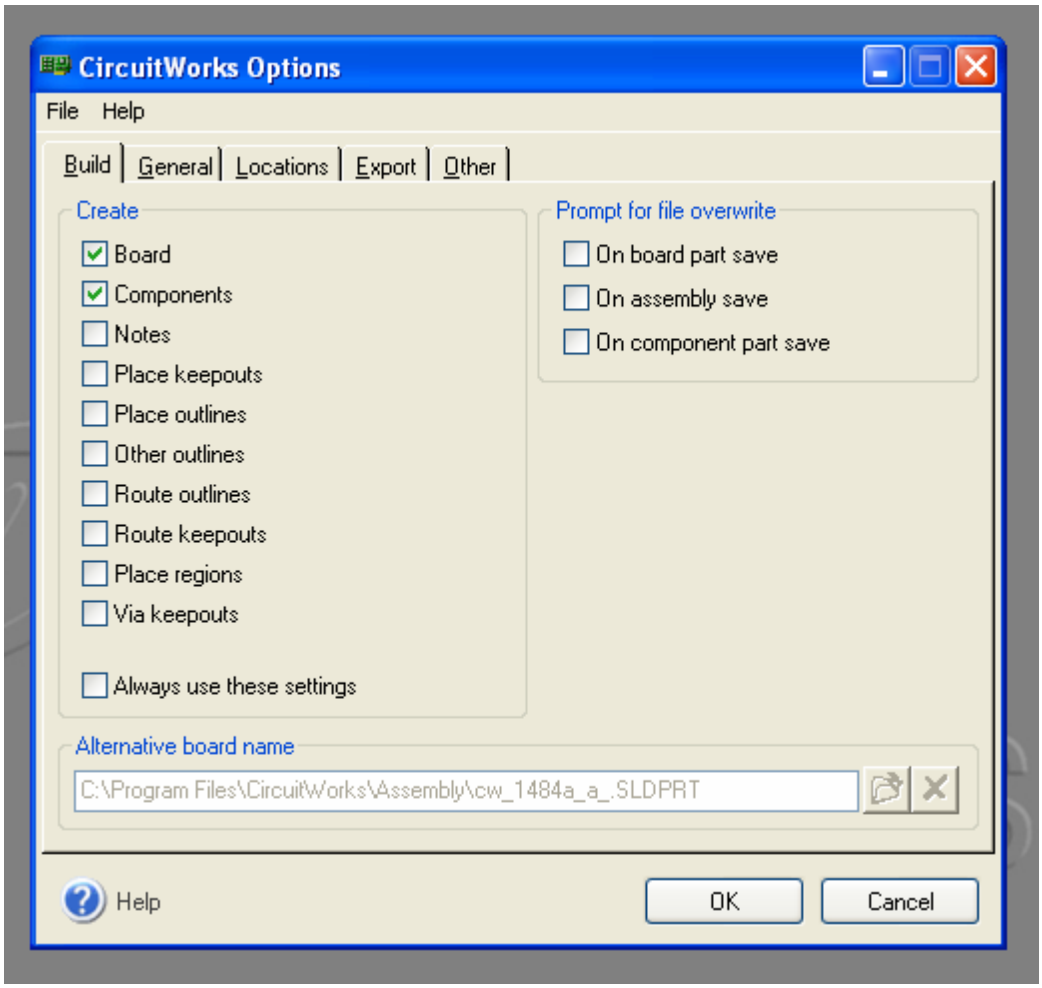
Show Reference Designators

When CircuitWorks creates a SolidWorks assembly, the part names of the components are displayed in the SolidWorks Feature Manager as usual. However, by selecting 'Show Reference Designators' from the *Board* menu, the Feature Tree display can be updated to reflect the reference designators of the components rather than their names. Selecting *'Show Component Names*' will set the Feature Tree back to its initial state.

CircuitWorks Options

The CircuitWorks Options dialogue provides a single point to set a wide variety of settings in CircuitWorks according to the preferences of the user, the type of files CircuitWorks is being used with, and the locale CircuitWorks is being used in.

To access the CircuitWorks Options dialogue, select *Tools / Options…* under the *CircuitWorks* menu in SolidWorks. The options dialogue will be displayed:



The CircuitWorks Options dialogue is separated into 5 separate pages, each accessed by a separate tab at the top of the dialogue.

Build Options

The settings on the build options page determine what features CircuitWorks builds in SolidWorks by default, and they are built. The options grouped together under the heading ‘Create’ determine what parts of the IDF file CircuitWorks builds in SolidWorks.

Board
With this option checked, CircuitWorks will create a new board part from scratch each time an IDF file is processed. If the option is unchecked then CircuitWorks will use the file specified in the ‘Alternative Board Name’ option at the bottom of the dialogue. If no alternative board name is specified, CircuitWorks will attempt to use a part file with the same name as the IDF file. If no suitable board part can be found an error message will be generated.

Components
Check this option to add the components to the board assembly.

Notes
Check to add any notes in the IDF file to the SolidWorks assembly

Place Keepouts, Component Keepouts & Via Keepouts
Check this option to add the component keepout areas to the board assembly. Keepouts are specific regions on the board where components cannot be placed. The keepout areas are modeled as a translucent part in the assembly.

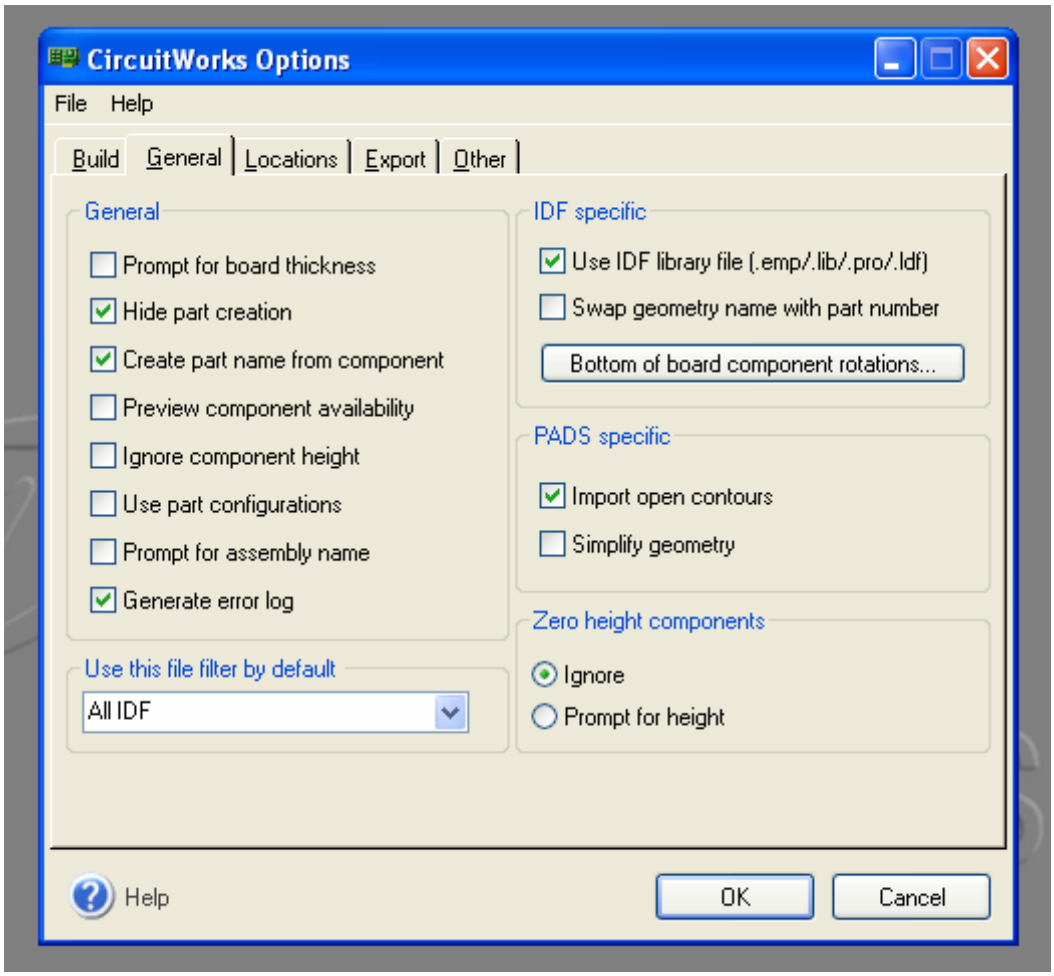
Place Outlines, Component Outlines & Other Outlines
Check this option to add the component outline areas to the board assembly. Outlines are specific regions on the board where components must be placed. They are defined as sketches in the completed assembly.

When ‘**Always use these settings**’ is checked, CircuitWorks will use the settings set above each time a board is processed. If this option is not checked, CircuitWorks will default back to only building the board and component models each time SolidWorks is restarted.

Prompt for file overwrite
When these options are checked, CircuitWorks will prompt the user before overwriting any existing files (normally as a result of the same board having been processed previously with CircuitWorks).

General Options

The second Tab on the Options dialogue groups together general CircuitWorks options, as well as options specific to the way CircuitWorks processes the IDF and PADS file formats.



Prompt for board thickness

Checking this option will allows the user to change the thickness of the Solid Model of the Printed Circuit Board. By default, CircuitWorks uses the board thickness defined in the IDF file, but checking this option will cause a dialog box to appear during the build process prompting the user for the required thickness.

Hide Part Creation

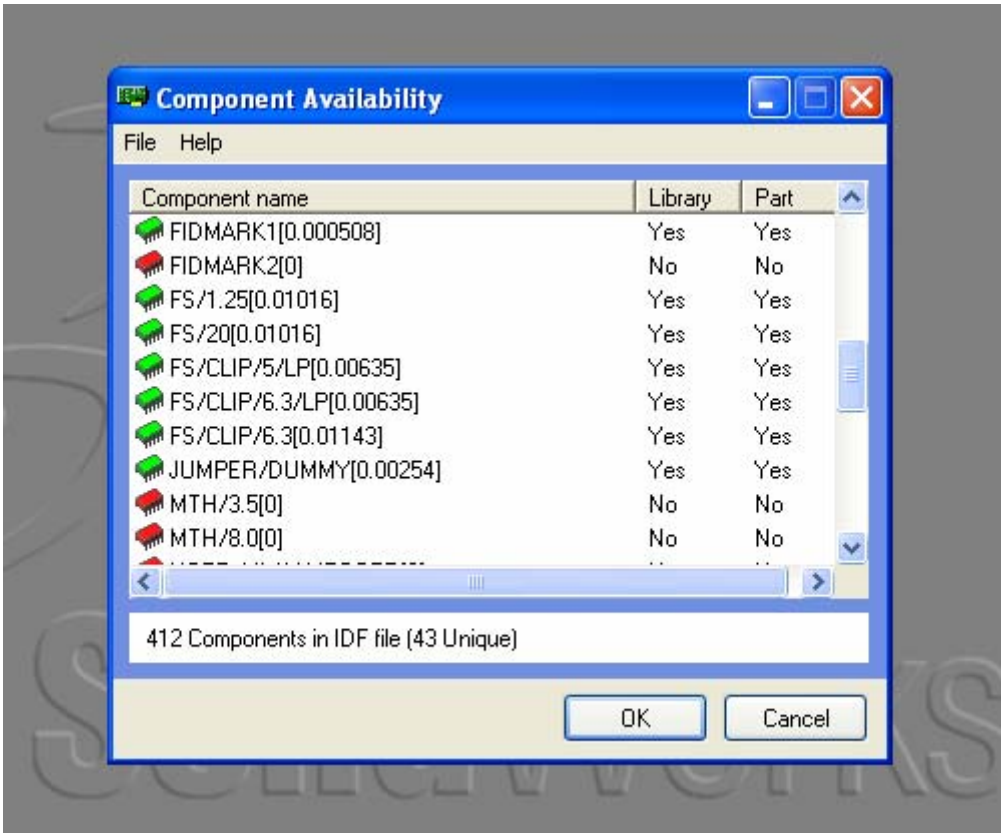
When this option is unchecked, the creation of any new component models in SolidWorks is visible to the user. Checking this option hides the build process which greatly increases the performance of CircuitWorks.

Create Part Name from Component

When this option is checked, new component models created by CircuitWorks will be named as closely as possible to their part names in the IDF file. With this option unchecked, CircuitWorks will name any component models with a sequential number. It is recommended this option is checked, so CircuitWorks names SolidWorks models in such a way they can be easily identified by the user.

Preview Component Availability

When this option is checked, CircuitWorks will display the Component Availability dialogue before as soon as an IDF file is opened, before starting the Import Wizard:



The Component Availability dialogue lists all the components that are in the IDF file with their heights (in meters), and indicates if they have been encountered before. If the icon is Green, CircuitWorks has encountered that component previously, and already has a SolidWorks model available for it. If the icon is red, either no record exists of the component in the CircuitWorks library, no Solid Model of the component exists, or both.

This dialogue can be used to determine how much time a board will take to build. If most of the components already exist (mostly green icons), the build will be built fairly quickly. The more red icons there are, the more Solid Models SolidWorks will have to build, so the slower the build process will be.

Clicking ‘Cancel’ will close the dialogue and cancel the build process. Clicking ‘OK’ will close the dialogue and continue with the Import Wizard.

Ignore component height

Normally, CircuitWorks uses a combination of both a component's name and its height as defined in the IDF file to determine if it has already been encountered by CircuitWorks. Checking this option will set CircuitWorks to match components purely on component name alone.

In most cases, the state of this option will make no difference to the operation of CircuitWorks, however some ECAD systems will define one component as having several different heights – a good example is a resistor mounted on a board on end, or flat – the same component but two different heights. With this option unchecked, CircuitWorks will produce two Solid Models to represent the two instances of the part, with the option checked; CircuitWorks will produce just one Solid Model and use it in both cases.

Use Part Configurations

With this option checked, CircuitWorks will use configurations when building the assembly. If one or more parts in the component library have been set to use anything other than their default or 'last saved' configuration, then checking this option will cause CircuitWorks to scan through the assembly after it has been built and set the components to their correct configurations. If no multi-configuration parts are used in your assembly, it is recommended this option is turned off to increase the speed of CircuitWorks.

Prompt for assembly name

When this option is checked, CircuitWorks will prompt the user for an assembly name to save the assembly once it has processed the IDF file. If this option is not checked, CircuitWorks will save the assembly with the same name as the IDF file in the location specified on the 'Locations' page of the Options dialogue.

Generate error log

If you are encountering technical problems processing an IDF file, checking this option will creates an error log file with the name **<boardname>.log** file in the **<data>** directory, detailing any errors that CircuitWorks found while processing the file (The **<data>** directory is defined in the *Locations* page of the Options dialogue).

This error file can be used to help your reseller diagnose any problems you are having.

Note: It is recommended that this option is unchecked unless you specifically want to generate an error log, as having it switched on will slow CircuitWorks down.

Use this File Filter by Default

IDF files have a number of suffixes which vary according to the system that produced them. Setting this option defines the default file extension for the open file dialogs in CircuitWorks. The default setting, 'All IDF' will list all files with known IDF file extensions.

IDF Specific Options

The following Options only apply when CircuitWorks is processing IDF rather than PADS files:

Use IDF library file

In normal circumstances, CircuitWorks requires both parts of IDF files to be present (normally *.emn/*.emp). The first part contains information to allow CircuitWorks to build the model of the board and position the components on the board; the second part of the file (the library file) contains information about the size and shape of the components on the board.

With this option unchecked CircuitWorks will not attempt to reference the IDF library file if it is present. However, if this option is unchecked without having a pre-built component part library available then you will find that CircuitWorks will display numerous missing part warning messages, as it does not have sufficient information to build the component models.

Swap geometry name with part number

By default, the component part names CircuitWorks gives to the component models it generates are derived from the geometry name defined in the IDF file. Checking this option will generate part names using the part number information defined in the IDF file instead.

For example, given the following placement entry in an IDF file:

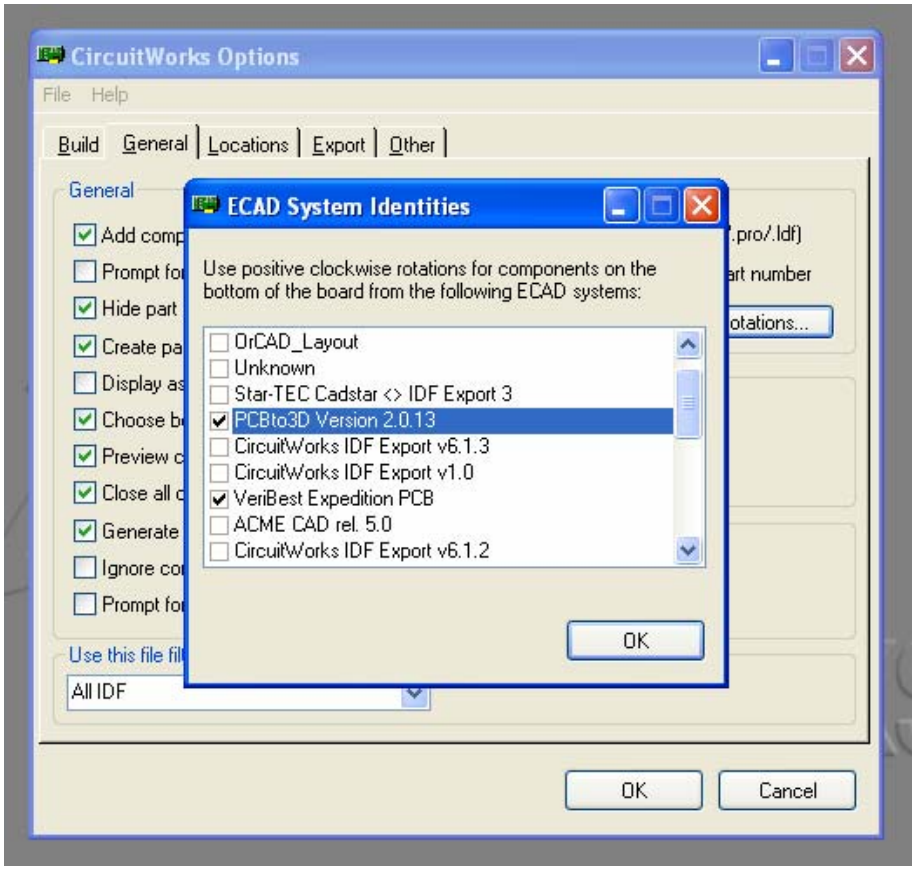
"15K0/B" "L14006" R4
139,6 59,4 0 0 TOP PLACED

Unchecked, this option will create a part name called **15K0B** (the / character will be removed as it's not valid in a SolidWorks filename)**,** while checking the option will create a part name **L14006**

Bottom of Board Component Rotations

Some ECAD systems misinterpret the IDF specification when rotating components on the underside of the board. Some systems will rotate components clockwise while others will rotate them anti-clockwise.

CircuitWorks attempts to learn which ECAD systems rotate components in which direction by building up a list of ECAD systems it has encountered. Clicking the 'Bottom of Board Rotations' button will display the following dialogue:



This dialogue lists all the ECAD systems that this particular seat of CircuitWorks has been used with so far, and whether to rotate the underside board components in the opposite direction.

Click on the check box next to the appropriate ECAD system to rotate underside components in the opposite direction.

PADS Specific Options

These options are specific to the PADS (*.asc) file format.

Import Open Contours

Unlike the IDF format, the PADS format is not specifically designed for import into Mechanical CAD systems. As a result, contours defined in the PADS file may not be closed, and so won't extrude properly in SolidWorks.

If this option is enabled then CircuitWorks will import all the entities that make up that particular component profile, even though they will most probably create an invalid sketch that will require some manual user editing to produce a sketch suitable for extrusion.

Simplify Geometry

As the PADS format is not specifically designed for Import into 3D CAD systems, it is possible to define profiles in PADS that are not suitable for extrusion in SolidWorks, open profiles and nested profiles for example. With this option checked, CircuitWorks will attempt to simplify the profile so it can be extruded in SolidWorks. It does this by looking at the minimum and maximum values for points in the profile and creating a simple sketch between those points and extruding the resulting profile. As a result, some detail may be lost from the resulting sketch.

Zero Height Components

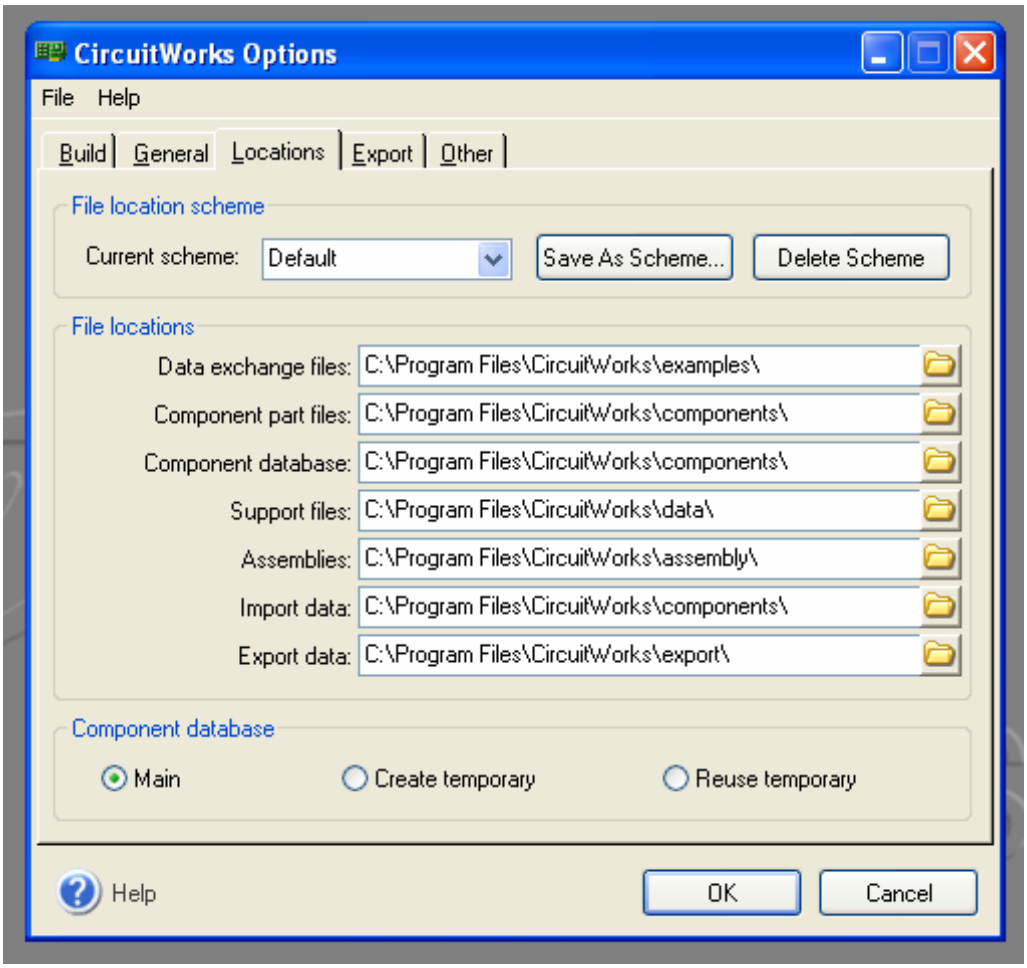
If this option is set to ‘Ignore’, then any components in the IDF file that don’t have a height defined, or have a height of zero will be left out of the completed assembly in SolidWorks.

If this option is set to ‘Prompt For Height’, CircuitWorks prompt the user for a height for the component every time a zero height component is encountered. As CircuitWorks builds up a library of components, the user will only be prompted when CircuitWorks encounters a new component. If component heights aren’t defined in your ECAD system, this option can be a useful and quick way to enter component heights into CircuitWorks.

An alternative to the use of this option is to edit the component heights in the ‘Component Heights’ page of the CircuitWorks Import Wizard, where zero height components are indicated with a red icon.

Locations Options

The locations page of the options dialogue is used to set the default locations CircuitWorks uses to store files and data, and to set how the database that controls its component library is managed.



File Location Scheme

By default, CircuitWorks has one file location scheme – one set of paths that defining the locations of data, components etc. However, it is possible to add one or more additional schemes. This feature can be useful when working on different projects for example – one scheme can be defined for each project, and different paths defined for each project. Components created in one Project could be stored in a different folder to components created in another Project for example.

To use a file scheme, select one from the 'Current Scheme' pull-down list. To add a new scheme, click the 'Save As Scheme…' button and type the name of your new file scheme. The paths can then be changed as required for that file scheme.
To delete a file location scheme, ensure it is visible in the pull-down list and click the 'Delete Scheme' button. Note that the default file scheme cannot be deleted.

File Locations

The default file locations that CircuitWorks uses are displayed in the file locations area. To change one of the default file locations, click the folder icon to the right of the displayed path and browse to a new location.

The file locations are:

Data exchange files – The default location where CircuitWorks looks for IDF and PADS files.

Component Part files - The location that CircuitWorks saves the component models it builds. If you have a more than one seat of CircuitWorks on a network, then changing this location to a shared central location available to all users will mean that one users seat of CircuitWorks will have the other users' component models available to use.

Component Database – This is the location where CircuitWorks maintains the central database (a Microsoft Access file called 'circuitworks.mdb') that it uses to control the component library. This file normally resides in the same location as the Component Part files by default. If the Component Part files location is moved to a shared network location, it is recommended this file location is changed too so each seat of CircuitWorks shares a common library.

Support files – Every time CircuitWorks processes an IDF file, it saves a file with the same name and a *.cwi* suffix in the Support Files location. This file is used by CircuitWorks to provide information about a processed IDF file in SolidWorks such as reference designators and component positions when the original IDF file used to create the assembly is not available. There is normally no need to change this file location from the default.

Assemblies – When CircuitWorks creates an assembly, it must be saved before any components can be inserted into it due to limitations in SolidWorks. This location is used to save the assemblies before the components are inserted. Once components have been inserted into the assembly and CircuitWorks has finished processing the IDF file, clicking the 'Save' icon in SolidWorks will save the completed assembly to this location. However, there is no need to save the assemblies CircuitWorks creates with any particular name, or in any particular location. The assembly location is only required so CircuitWorks can insert components into the assembly.

Import Data – This is the location CircuitWorks uses by default when importing data into the Component Library.

Export Data – This is the location used by default to save IDF files that CircuitWorks has created using either the Full Export or Express Export. In both cases the user can override this default setting and save the IDF files anywhere.

Component Database

CircuitWorks uses a simple Microsoft access database to manage the CircuitWorks component library. The database cross-references the component's name as it appears in the IDF file to the SolidWorks file to use to represent it.

Under normal circumstances, the main database option should be used; however it is possible to use a temporary database instead of the main database, perhaps if you're working on a project for a different user don't want to use your current component library.

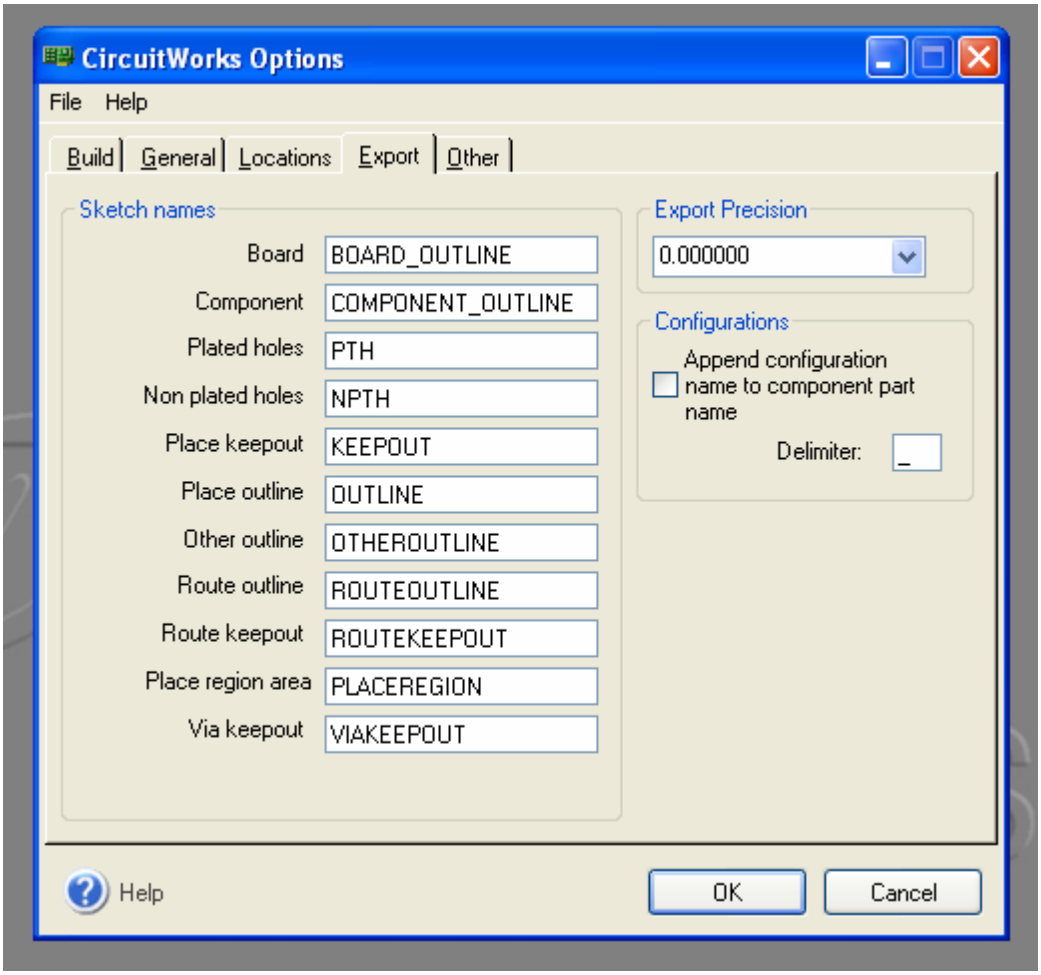
With the ‘Create Temporary’ option selected, CircuitWorks will create a new temporary component library each time it is started. Every time SolidWorks is restarted, the temporary database will be overwritten, so CircuitWorks always starts with an empty component library.

Selecting the ‘Reuse Temporary’ option will cause CircuitWorks to start a new temporary database like the ‘Create Temporary’ option, but unlike that option, the same temporary database will be used until the ‘Main’ component database is reselected, i.e., The temporary database will slowly grow in size like the Main database.

Note that after changing the database options, you must restart SolidWorks for the changes to take effect.

Export Options

The Export Options page of the Options dialogue controls Options that CircuitWorks uses when saving an assembly as an IDF file using the Full Export functionality.



Sketch Names

If you look at the feature tree in SolidWorks in an assembly that CircuitWorks has created, you'll notice that features such as the Board Extrusion, Holes and Keepouts are named in the feature tree. As well as making the assembly CircuitWorks creates easy to understand, these sketch names are used by CircuitWorks when exporting an assembly as an IDF file.

CircuitWorks uses the names of the features to determine what the features represent, which extrusion is the PCB, which extrusions are Keepouts, which are Components etc…

If you are exporting an assembly that CircuitWorks has created, then the features will be correctly named for export. However, if you are exporting an assembly that has been modeled directly in SolidWorks, then the features should be renamed according to the names displayed on this dialogue to ensure that the assembly exports correctly.

Any of the default sketch names can be changed as required.

Export Precision

This pull-down list sets the number of decimal places CircuitWorks exports to. It may need to be changed to suit certain ECAD systems.

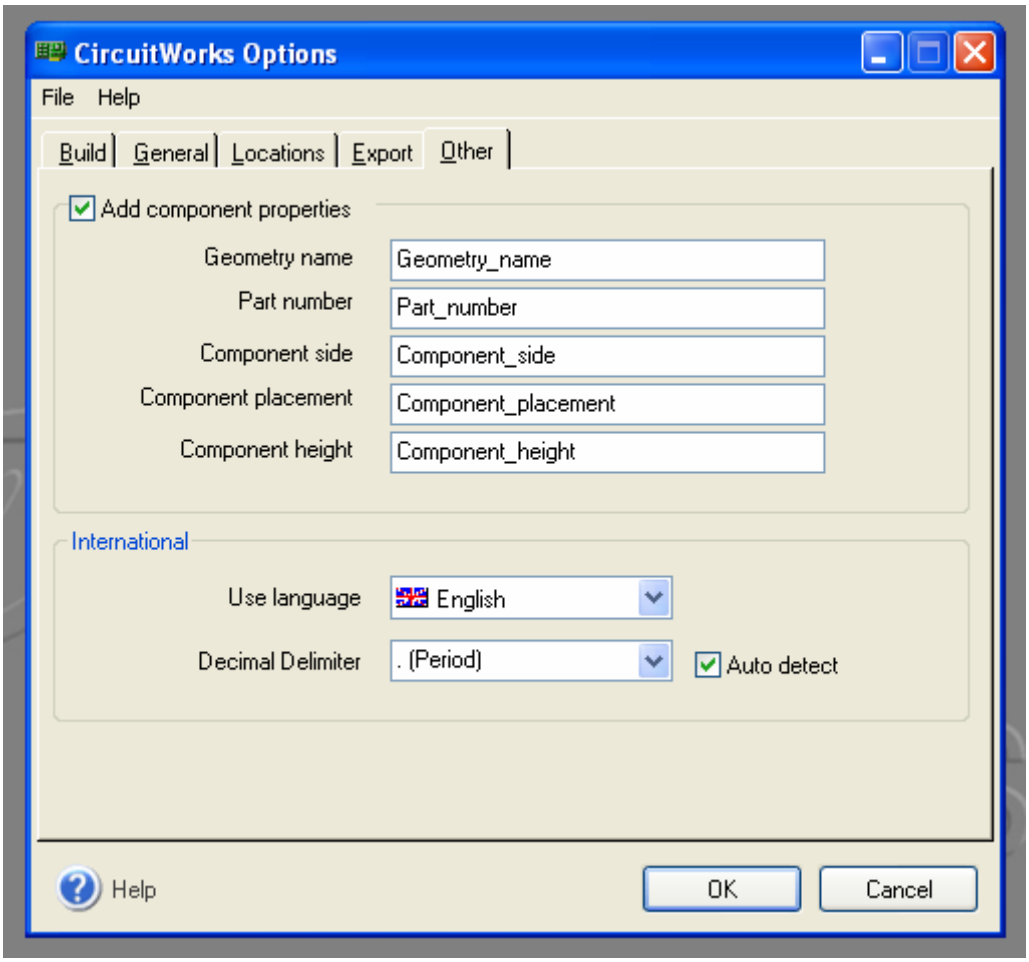
Configurations

When exporting an assembly, CircuitWorks uses the name of the component in SolidWorks as the name to be used for the component in the IDF file. Checking this option will append the name of the configuration to the SolidWorks component name using the specified delimiter.

For example, if a component was called ‘SMR_0603’ in SolidWorks with the active configuration called ‘Resistor’, then the component name in the exported IDF file would be ‘SMR_0603_Resistor’ if the delimiter was set as ‘_’.

Other Options

The ‘Other’ options tab in the CircuitWorks options dialogue is used to define any SolidWorks custom properties CircuitWorks writes into newly created component models, and to set options for International users.



Custom Property Names

Checking this option will add the following custom properties to each component created by CircuitWorks. The custom property names can be changed via text boxes to suit predefined BOM templates, drawing borders or an existing PDM system. The default custom property names are:

Part number,i.e. X-SMO
Geometry name,i.e. XTAL CHIP 7
Component side,i.e. TOP
Component placement,i.e. PLACED
Component height,i.e. 0.00254 (specified in metres)

All the custom properties are of the type “text”, except component height, which is written as a number. The IDF 3.0 format supports properties that are defined in the ECAD system. Any Component Property Records found in the ECAD system will also be added to the components custom properties, e.g. “Capacitance”, “0.0025uF”

International Settings

Language

Use the Language pull-down list to change the language currently used by CircuitWorks.

Decimal Delimiter

By default, CircuitWorks uses a period ‘.’ to indicate a decimal delimiter. Some countries, such as France, use a comma ‘,’ to indicate a decimal delimiter. CircuitWorks will attempt to set this option automatically if the ‘Auto Detect’ option in checked, but it can be overridden by un-checking the ‘Auto Detect’ option and manually setting the Decimal Delimiter option.

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Supported Data Exchange Formats

CircuitWorks 7 supports the following data exchange formats:

IDF 2.0
IDF 3.0
PADS-ASCII PowerPCB 3.0 and above

CircuitWorks 7 feature comparison

		IDF 2.0	**IDF 3.0**	**PADS**
	Data exchange file extension support	EMN & EMP	EMN & EMP	ASC
		BRD & LIB	BRD & LIB	
		BDF & LDF	BDF & LDF	
IMPORT	Board	Yes	Yes	Yes
	Board cutouts	Yes	Yes	Yes
	Board drilled holes	Yes	Yes	No
	Board keepouts	Yes	Yes	No
	Board outlines	Yes	Yes	No
	Components	Yes	Yes	Yes
	Board drilled hole filtering	Yes	Yes	No
	Cutout filtering	Yes	Yes	No
	Component filtering	Yes	Yes	Yes
EXPORT	Board	Yes	Yes	No
	Board cutouts	Yes	Yes	No
	Board drilled holes	Yes	Yes	No
	Board keepouts	Yes	Yes	No
	Board outlines	Yes	Yes	No
	Components	Yes	Yes	No
	Notes	No	Yes	No
	Date exchange file custom attributes	No	Yes	No

About Priware Limited

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