

### Procedure Description

How to create and use import assignments

### Requirements

CircuitCAM software

Knowledge of the CAD package you are working with

### Procedure Solution

*What is an Import Assignment?*

An import assignment is a way to allow the automation of the importation process of your Gerber and NCDrill files, to allow for customization of the CircuitCAM software for various CAD package file types.

*How do I create my own import assignments?*

The first step in creating import assignments, is to know the format that your CAD package outputs its files in. Questions to ask are:

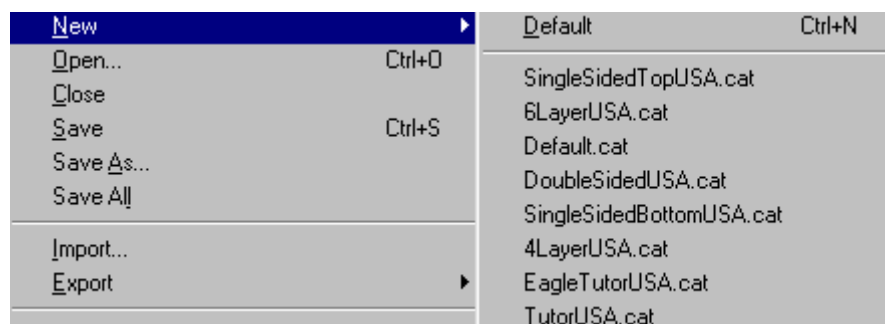
- What are the file extensions in use?
- What are the file prefixes in use?
- How does my CAD package designate the layer names?
- What is the output format of my CAD package?

When you have answered these questions, you will be ready to work on setting up your import assignments.

*What is the first step I should take?*

### Choosing a starting Template

The first step you should take, is to open the CircuitCAM software, and from the file menu, choose a New Template that best describes the type of circuit board you will be working with, such as, Single Sided, Double Sided, or even a Four Layer template file.



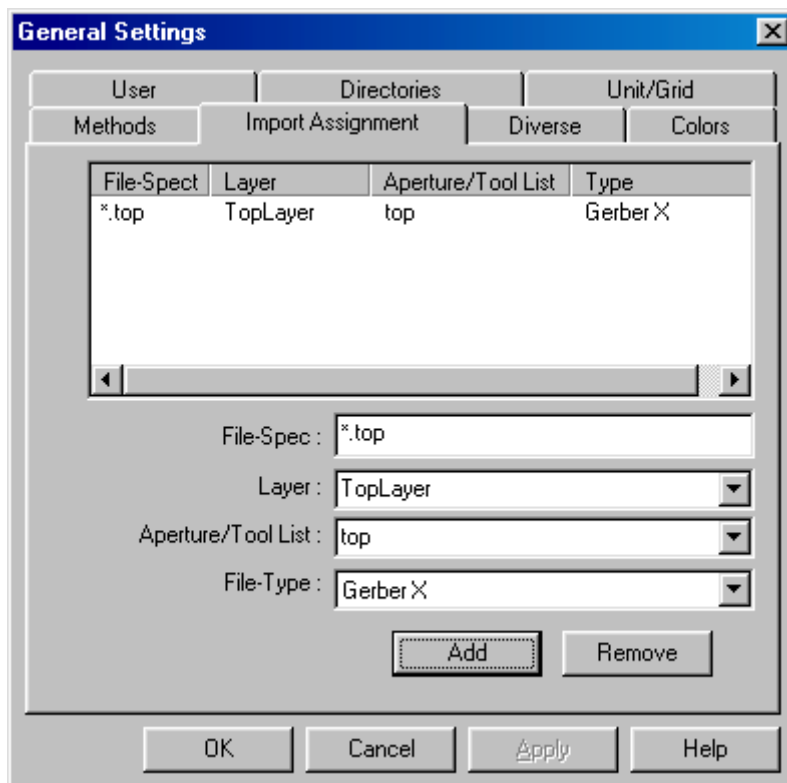
### Opening up the Import Assignment Dialog

The Next step to take is to go to the *Config* menu, and choose the *General Settings* option. In this dialog box, you will want to select the tab labeled *Import Assignments*. In this dialog box, you will be basically duplicating the functions you would normally perform manually, such as, selecting the file type, the destination layer, etc. The following is the list of steps to take, in order of entry.

### Gerber X

If you are importing **GerberX** files, use the following setup information.

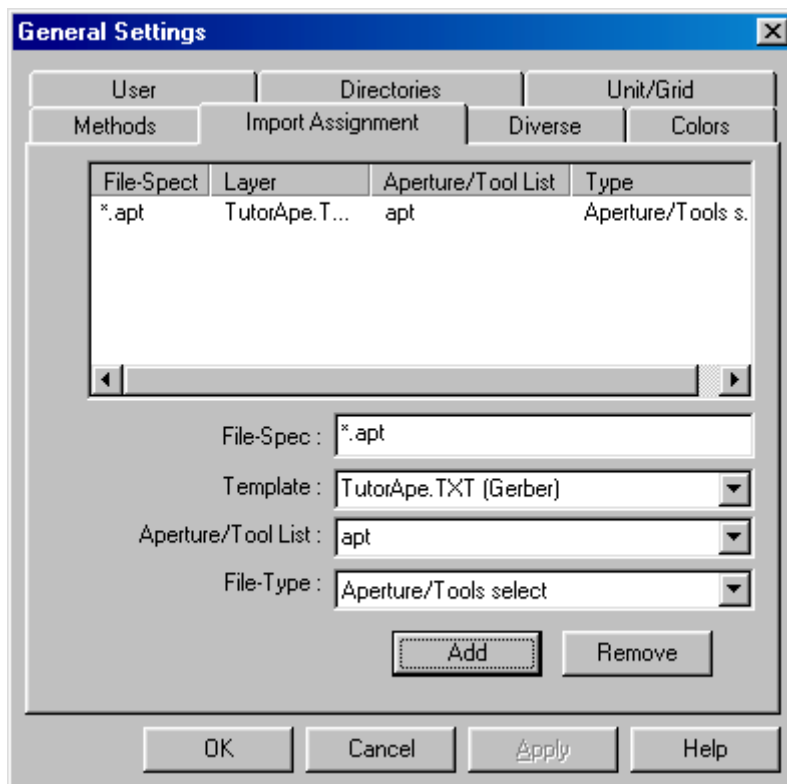
- In the **File-Type** drop down box, choose *Gerber X*.
- In the **Aperture/Tool List** box, enter in the name of the aperture list, which will be contained in the Gerber X file, preferably one that reflects the layer that you are going to be importing.
- In the **Layer** box, choose the destination layer for the file you are going to import.
- In the File-Spect box, you would enter in the name of the file you are going to import, using wildcards (\*) for the characters that will change with each project.



### Gerber (non Gerber X)

#### Aperture Files

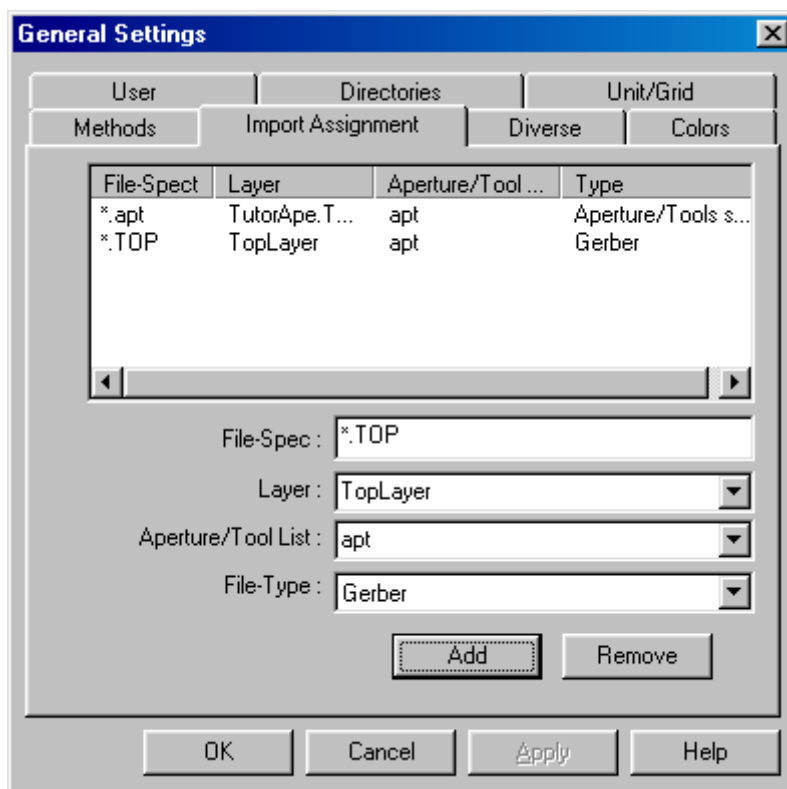
- If you are importing an Aperture file, you will want your **File-Type** to be *Aperture/Tools select*.
- In the *Aperture/Tool List* box, enter in a name for your Aperture list you are about to import, preferably one that reflects what layer you are defining
- Next, select the *Template* of the type that is used to read in the Aperture list of your CAD package.
- For Aperture files, define the *File-Spec*, or, the mask of the file that you are going to import, using a *wildcard (\*)* for any characters that may change for each different projects, and enter in the prefix or the extension of the file that will always remain the same.



### Gerber Files

When importing a Gerber file, you need to have predefined an aperture list to use with the Gerber file. After defining the Aperture list, you may read in the Gerber file by the following procedure.

- In the **File-Type** drop down box, select *Gerber*.
- In the **Aperture/Tool List** box, select the name of the previously defined aperture, in our case here, being *apt*.
- In the **Layer** box, select the intended destination layer for your imported file.
- For the **File-Spec**, select the name of the file to import, using *wildcards (\*)* to replace any characters that will change from project to project.

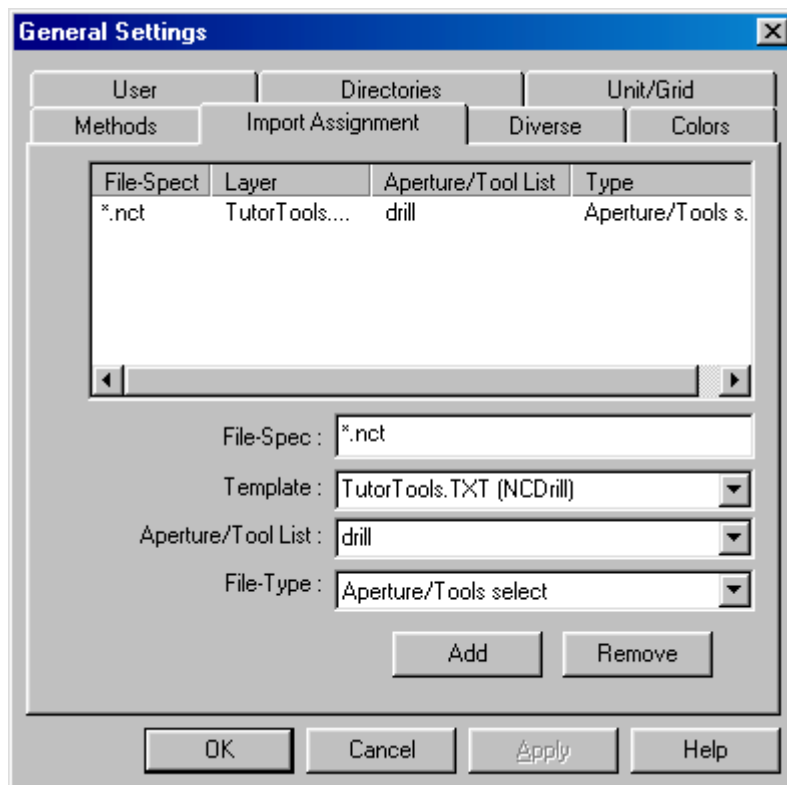


### NCDrill Files

When you are going to set up the import assignments for NCDrill files, there are two types of files you may be working with. The first is called *Excellon1*, and the second is called *Excellon2*. The difference is, that Excellon1 files need to have a separate Aperture/Tool list read in for it, whereas Excellon2 has an embedded Aperture/Tool list, just like *Gerber* and *GerberX*.

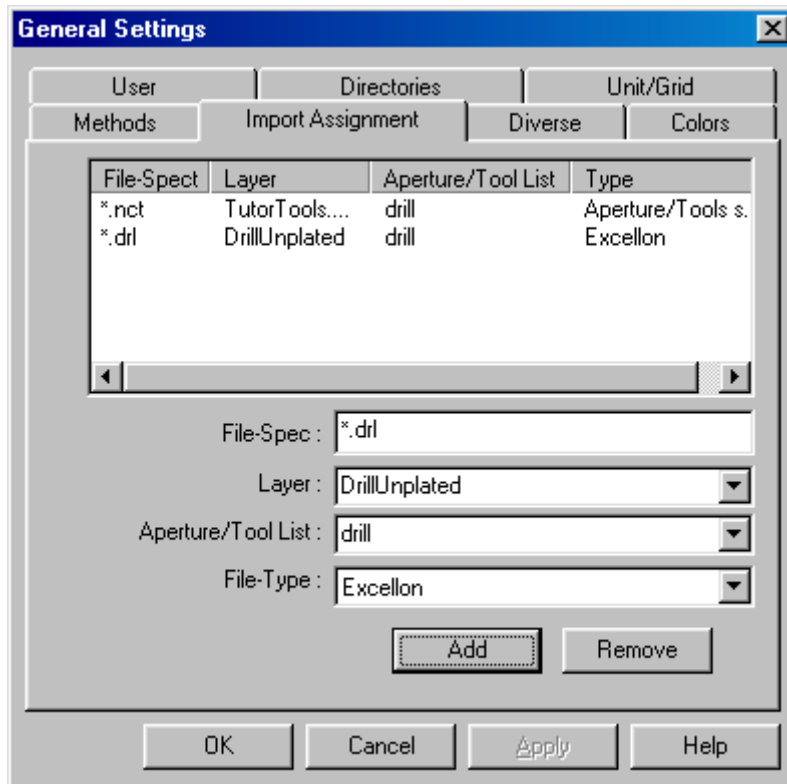
#### **Excellon1**

- **Aperture/Tool List**
  - To define your *Aperture/Tool list* file, you will want your **File-Type** to be *Aperture/Tools select*.
  - In the **Aperture/Tool List** box, enter in a name for your Aperture list you are about to import, preferably one that reflects what layer you are defining
  - Next, select the **Template** of the type that is used to read in the Aperture/Tool list of your CAD package.
  - For Aperture files, define the **File-Spec**, or, the mask of the file that you are going to import, using a *wild-card(\*)* for any characters that may change for each different projects, and enter in the prefix or the extension of the file that will always remain the same.



## How-To-Document

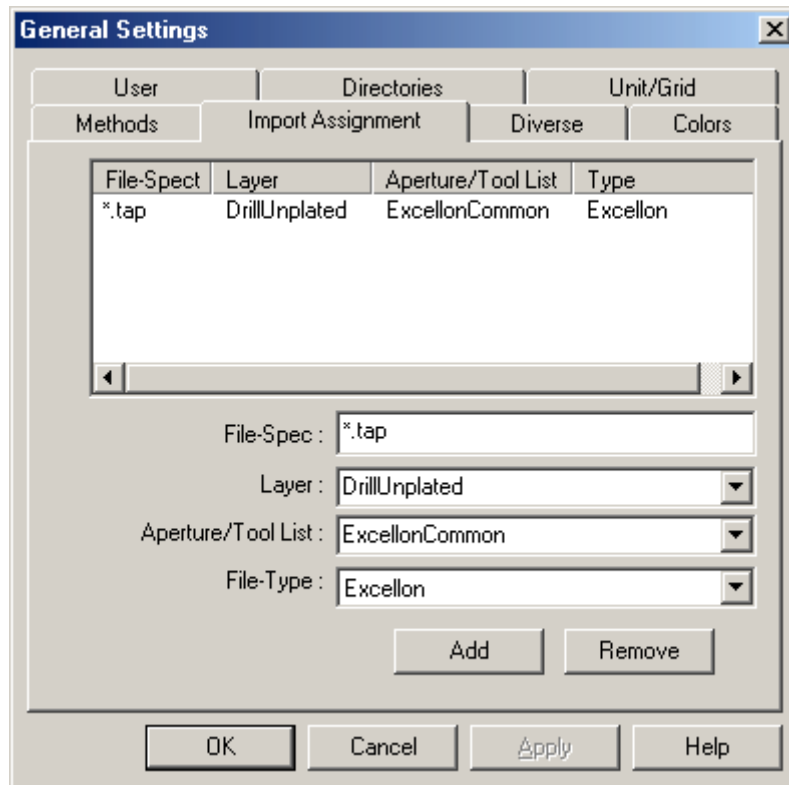
- **NCDrill File (x and Y coordinates)**
  - In the **File-Type** drop down box, select *Excellon*.
  - In the **Aperture/Tool List box**, choose the name of the previously defined tool list.
  - In the **Layer** box, select the name of the destination layer.
  - In the **File-Spec** box, type in the name of the file you intend to import, using a *wildcard* (\*) for any characters that will change from project to project.



### Excellon2

For Embedded Aperture/Tool list Excellon files follow the following procedure.

- In **File-Type** select *Excellon*.
- For the **Aperture/Tool List** choose *ExcellonCommon*.
- In the **Layer** box, select the appropriate *drill layer*.
- In the **File-Spec** box, enter in the name of the drill file you are going to import, using *wildcards(\*)* for any characters that will change from project to project.



### References

CircuitCAM 3.2 manual  
CircuitCAM 3.2 software

### Author

Craig Kniskern