

Read Me

IEEE 315-1975 Symbol Library

Sample Part



The IEEE 315-1975 Library Sample Part is in a zip folder. Open the folder by clicking the link within the email sent to you by the Virtual Interconnect Shop. Save the files in the folder to your desired location.

What's included

The sample part includes the following files:

Name	Type	Description
9_1_1.sheets	Sheets File	Creo Schematics design sheets file
fuse 9.1.1	PKG File	Creo Schematics export package file
fuse 9.1.1-contents	MASM Listing	Structure of the export package
Library_full_global_props	CSV File	Global Properties file

Viewing the sample in Creo Schematics 2.0

To view the sample in Creo Schematics 2.0 use the Import > Sheets function:

1. Create a New Design in Creo Schematics. **File > New > Create New Design**. Give your design a suitable Name and Description and ensure Use Template is unchecked.
2. **Close** the Design Explorer.
3. **File > Import > Design Sheets**.
4. Browse to the 9_1_1.sheets file and select it as the file to import. **Next > Next > Import > Finish**.
5. Open the Design Explorer and the sheet.
6. The sample symbol is included in the sheet and it is also available to instance from the Catalog Explorer.

Viewing the sample in Creo Schematics 1.0

The sample part is in a package file format. To view the sample you must import it into your design:

1. Open Creo Schematics.
2. Create a new design: **File > New > Create New Design**. In the **New Design** dialog box, give your design a Name and Description and choose a Location to save it, as below. Click **OK**.



Figure 1: New Design dialog box

3. The **Design Explorer** opens. **Create a New Sheet** . Choose the desired Design Template then double click on the sheet name to open it. **Close** the Design Explorer.

You now need to import the package into your design.

4. Click **File > Import > Package**. The **Import Package Manager** dialog box opens.

Read Me

IEEE 315-1975 Symbol Library

Sample Part

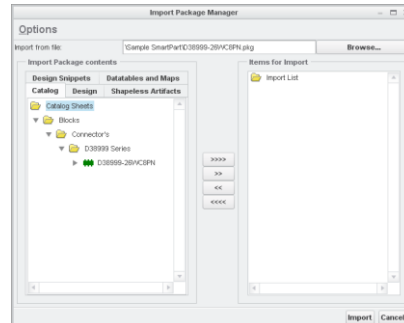


Figure 2: Import Package Manager dialog box

5. Click **Browse** and browse to the location of the **fuse 9.1.1.pkg** file.
6. Select the package (PKG) file and click **OK**.
7. Click **>>>>** to Import All the items in the list.
8. Click **Import**. When the objects being imported already exist, the Import Package Comparator dialog box opens.
9. Right click and choose **Select all IMPORTED attributes** to accept all the attribute values from the imported package.

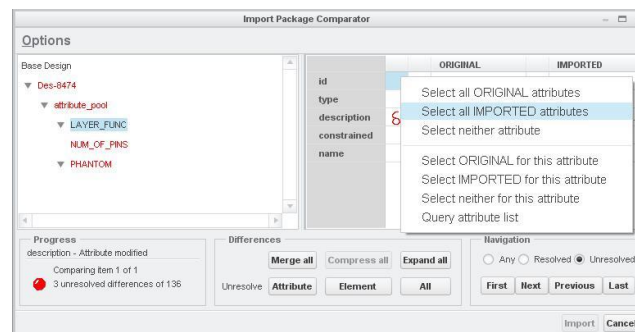


Figure 3: Import Package Comparator dialog box

10. Click **Import**.

The sample part is now available to instance from the Catalog Explorer.



11. Open the Catalog Explorer by selecting **Diagramming > Catalog Explorer**.
12. Change the Diagram type to All Types. You can now Instance the Wiring or Circuit symbol onto the Design Sheet.
13. To view the properties of the component, Right click on its Name in the Catalog Explorer and select **Properties**.

Read Me

IEEE 315-1975 Symbol Library

Sample Part



Setting the Global Parameters

If you wish to import the global parameters from the sample part into your design, use the CSV file included and follow these steps:

1. Click **Admin > Edit global parameters**. The **Define Types** dialog box opens.
2. Click **Import** and navigate to the **Library_full_global_props.csv** file. Click **OK**. **Close** the **Define Types** dialog box. The global properties are now loaded into the design.

Support

If you have any problems importing the sample or would like any further information on Virtual Interconnect's range of component libraries for Creo Schematics, please do not hesitate to contact us:

Email	info@virtual-interconnect.co.uk
Telephone	+44 (0)141 530 5567

Read Me

IEEE 315-1975 Symbol Library

Sample Part



Notes

1) Crimp/Terminals

Note that every port of a connector has the property “term_name”, which is “UNSET”. This is to permit the assignment of a terminal part number, which can be included in BoM reports.

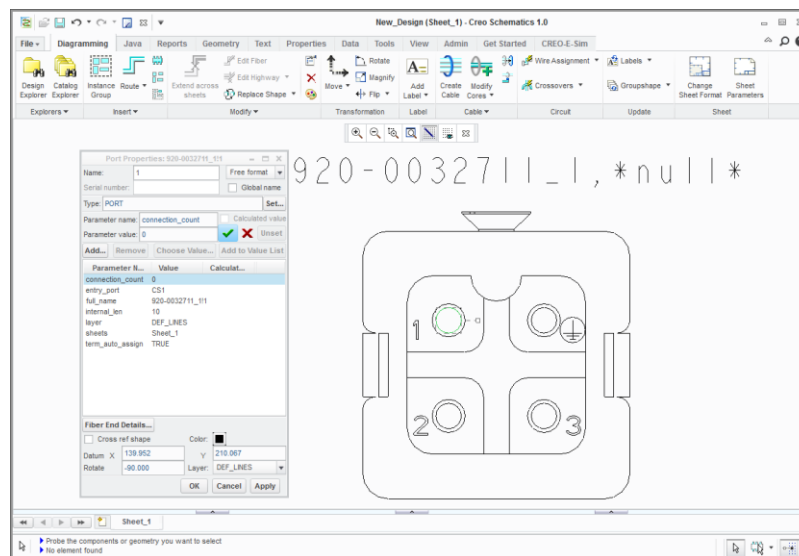


Figure 4: Port Properties

2) Reference Designator

Note that each connector has labels attached for the part name and the Reference Designator, which must be manually assigned for every instance of the connector e.g. J1, J2, J3 etc.

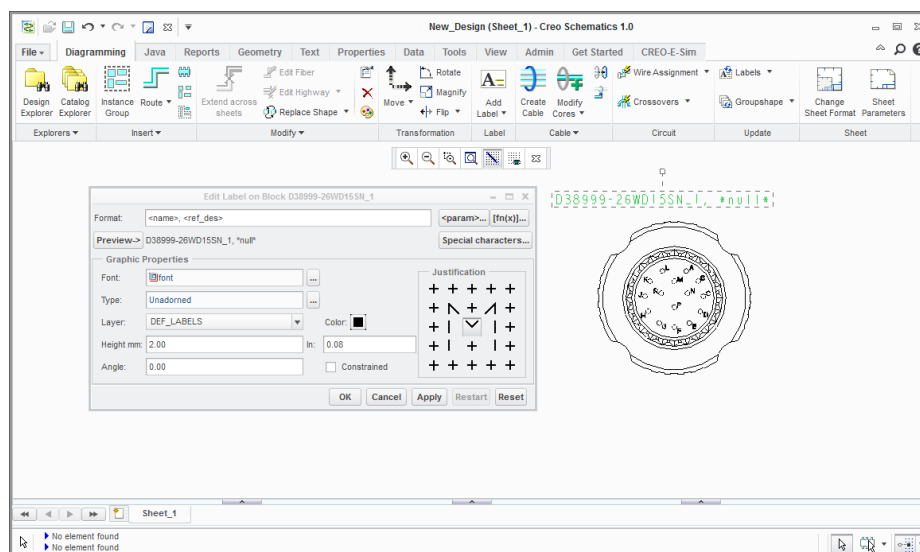


Figure 5: Reference Designators