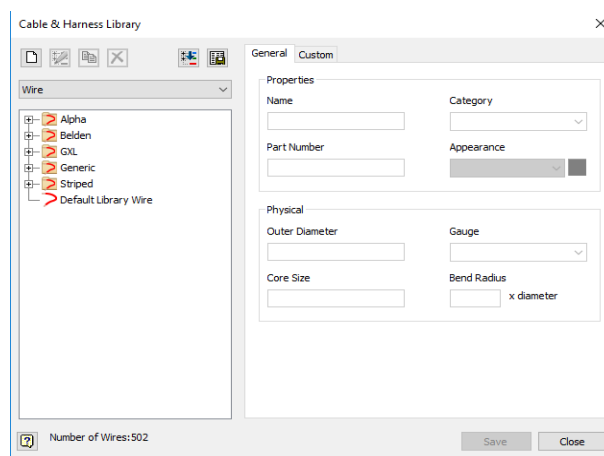# Defining Wire Definitions Within Inventor

Autodesk Inventor 2019

Wire type definitions must be defined correctly within Autodesk Inventor and match the wire types defined in AutoCAD Electrical if you wish the wire types to be created from the AutoCAD Electrical data, subsequently routed and displayed correctly within your assembly. Within the *Cable & Harness Library* dialogue, you can define new wire, cable, raw ribbon cable, splice or virtual part descriptions to the library or update existing library definitions.

> The Inventor wire *Part Number* **MUST** match the wire types (layer name) defined within AutoCAD Electrical. If they are slightly different Inventor will not recognise them and use a generic wire type instead for routing purposes.

You must an existing harness for the *Cable & Harness* ribbon tab to be displayed.

Select the *Cable & Harness* ribbon tab > *Manage* panel >

	Create a new wire definition		Deletes a wire definition
	Edits an existing wire definition		Imports Library objects
	Copies a wire definition		Exports Library objects

To copy an existing wire type:

The easiest method of creating a new wire type is to copy an existing similar one as you have less to edit.

Ensure | Wire ⌄ | is selected

Expand [+] *Generic*

Right click over 10AWK_BLK and *Copy*

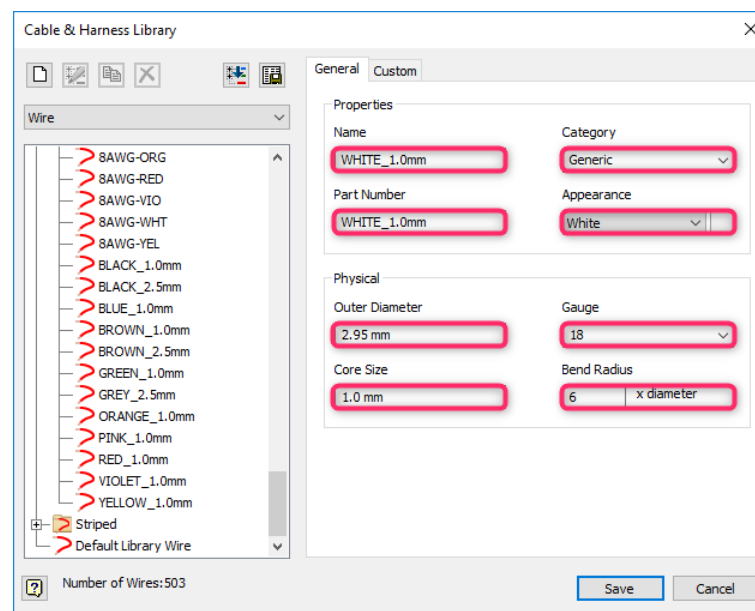*Right click* over the copied wire definition and *Edit*

Change the values accordingly

To create a new wire type from scratch:

Ensure | Wire ⌄ | is selected

Expand [+] *Generic*

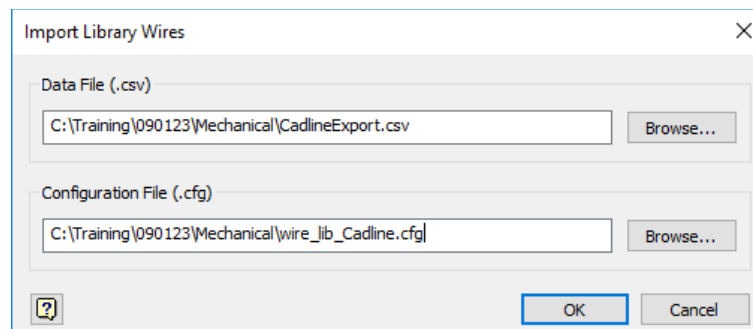Select ⬜ *New*

Enter the details accordingly:

Select Save

Select Close

You can also import an existing definition file. You can import new wire, cable, raw ribbon cable, splice or virtual part descriptions to the library or update existing library definitions. More information on what can be imported can be found at:

[Cable and Harness Library import configuration file format](#)

Select ⬇️ *Import Library Objects*

To import information from a csv file, you must have both a csv file and a cfg (configuration) file.

The cfg file is made up as follows (for wires):

libwires
c:\temp\log.txt
Wire Name,Appearance,Outer Diameter,Category,Part Number,Gauge,Color,Core Size,Bend Radius
text,text,length,text,text,text,text,length,unitless
none,none,mm,none,none,none,none,mm,none

The csv file is made up as follows e.g.:

BLACK_2.5mm,Black,3.3,Generic,BLACK_2.5mm,14,Black,2.5,6
WHITE_1.0mm,White,2.95,Generic,BLUE_1.0mm,18,White,1.0,6

Select OK

Select Close

You can also export the *Cable & Harness* Definitions to another Inventor installation (if not shared data).

Select 📊 *Export*