



# Defining a New Cable Within Inventor


## Autodesk Inventor 2021

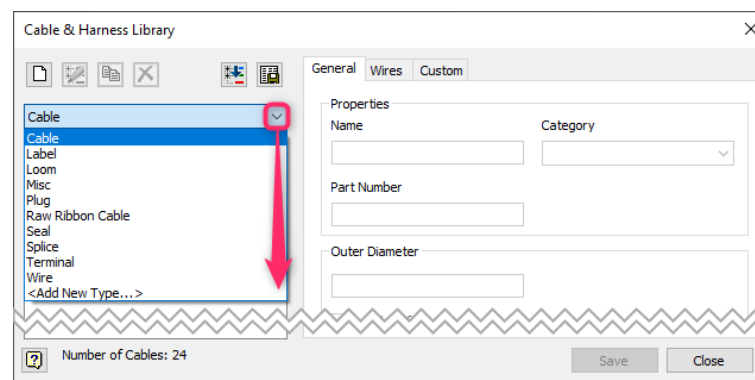
Cable type definitions must be defined correctly within Autodesk Inventor and match the cables defined in AutoCAD Electrical if you wish the cables to be created from the AutoCAD Electrical data, subsequently routed and displayed correctly within your assembly. Within the *Cable & Harness Library* dialogue, 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 cable *Catalog* number defined within AutoCAD Electrical. In addition, the order of the cores and the core identifications must also match. If they are slightly different Inventor will not recognise them.







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

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

Select the  arrow, as shown, to select *Cable*



### Cable options:

- |   |                                    |   |                            |
|---|------------------------------------|---|----------------------------|
|  | Create a new cable definition      |  | Deletes a cable definition |
|  | Edits an existing cable definition |  | Imports Library objects    |
|  | Copies a cable definition          |  | Exports Library objects    |

### Creating a cable type:

The core colours and sizes of a cable must exist as wires within the *Wire Library* before you can create a new cable. This means that cores cannot be simply defined as the core colours, but must have the gauge within the name for its definition. A cable must have at least one core (wire) in order to save/exist.

Ensure  is selected.

Select  New

Enter the details accordingly:

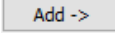
Properties	
Name	Category
<input type="text" value="4Cx2.5mm"/>	<input type="text" value="Generic"/>
Part Number	
<input type="text" value="4Cx2.5mm"/>	
Outer Diameter	
<input type="text"/>	
<input checked="" type="checkbox"/> Calculate from Wires	
Bend Radius	
<input type="text"/>	x diameter
<input checked="" type="checkbox"/> Calculate from Wires	

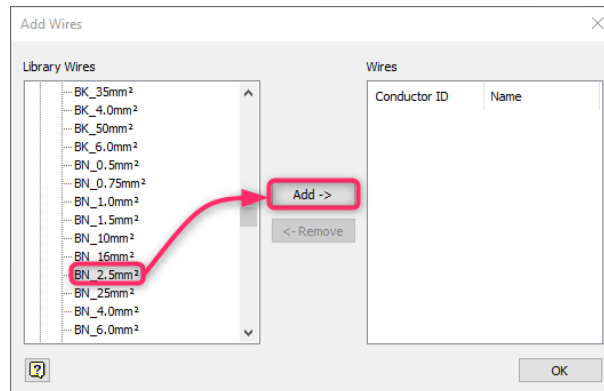
N.B. The Outer diameter can either be entered or calculated from the core (wire) bundle.  
The bend radius can either be entered or calculated from the core (wire) bundle.

Select the  tab

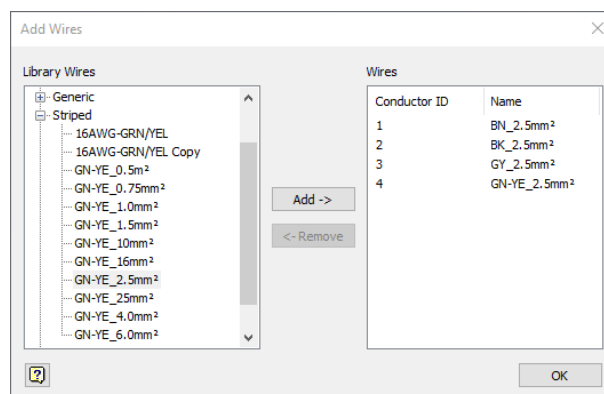
Select

Expand [+] the selected wire type


Highlight the wire type you wish to have as a core and select 



Repeat until all of the cores have been defined:



Select 

Select  tab if you wish to add your own custom properties.

Select 

**Note** the conductor names cannot be simply BN, BK, GY etc and must be unique. For Inventor to tally up with AutoCAD Electrical, the core identifications defined inside of AutoCAD Electricals' parts database (default\_cat.mdb) must match the core (wire name) definitions defined within Inventor. In addition, the conductor ID order must also match the order of the cores defined within AutoCAD Electrical if you wish to utilise the electromechanical integration functionality for cables.