


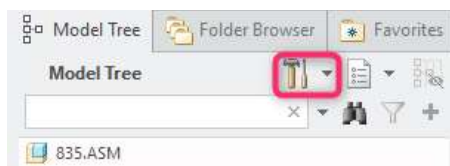
elecworks™ Tips & Tricks

Changing The Display Configuration Within elecworks™ for PTC Creo

The model tree occupies the left side of the screen of PTC Creo and shows all the sub-assemblies, parts, features, sketches, etc. of the main assembly. It is recommended to activate the “Display” “Features” option and show the columns “Designation” and “com_tag”.

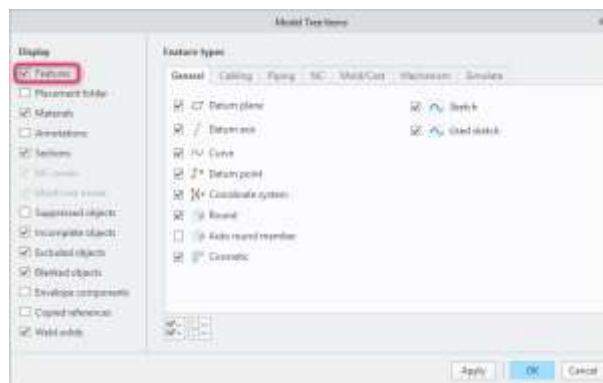
To show features:

On the *Navigation area*, select  *Settings* ↓



Select  *Tree Filters...*

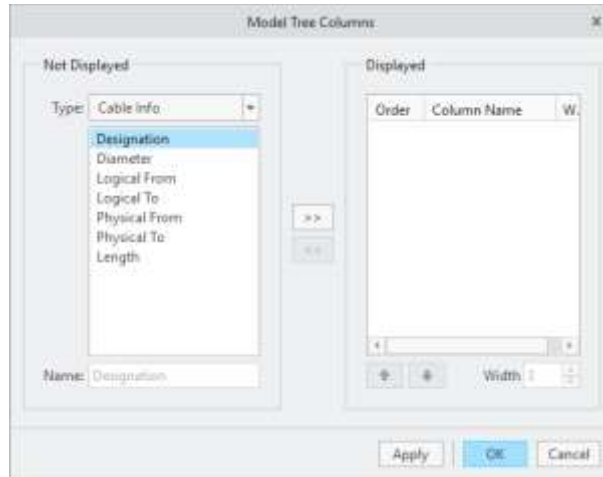
Ensure *Features* is selected



Select 

To show elecworks component information

On the *Navigation area*, select  **Settings** ↓ >  Tree Columns...

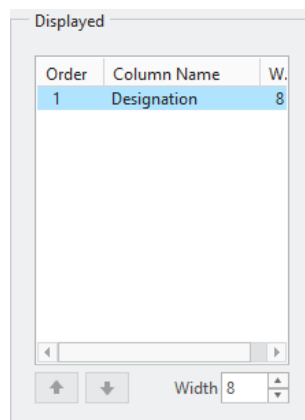


Select type

Highlight **Designation**

Select

The designation will then be shown to the right of the dialogue



Repeat the process:


Select type

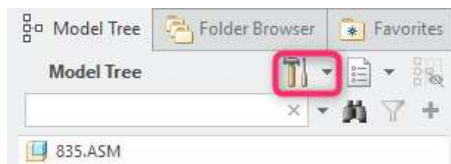
Type COM_TAG in the *Name* field


Select

Select

To save the settings:

On the *Navigation area*, select  *Settings* ↓



Select  *Save Settings File...*

Specify the file location for *tree.cfg*

Retrieving the display settings:

On the *Navigation area*, select  *Settings* ↓ >  *Open Settings File...*

Browse to the file location for *tree.cfg*

Highlight the file

Select