
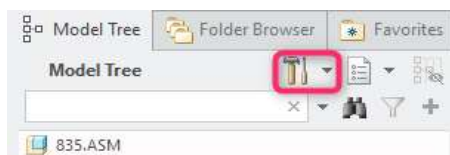



elecworks™ Tips & Tricks

Saving The Tree & Display Options Within PTC Creo

If you have created a display configuration file for your *Navigational area* palette, this can be saved and automatically loaded for each new assembly created thus avoiding the requirement to load on a per assembly basis. In addition the datum display filters of Axis display, Point display, Csys display and Plane display settings can also be saved.

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



Select  Save Settings File...

Specify the file location for *tree.cfg*

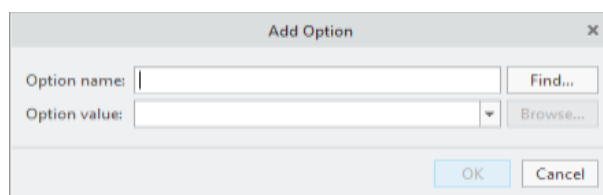
As an example the *tree.cfg* could be saved within the following directory:

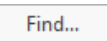
C:\Program Files\PTC\Creo #.#\M###\Common Files\text

Select the *File* ribbon tab >  Options

Select *Configuration Editor*

Select 



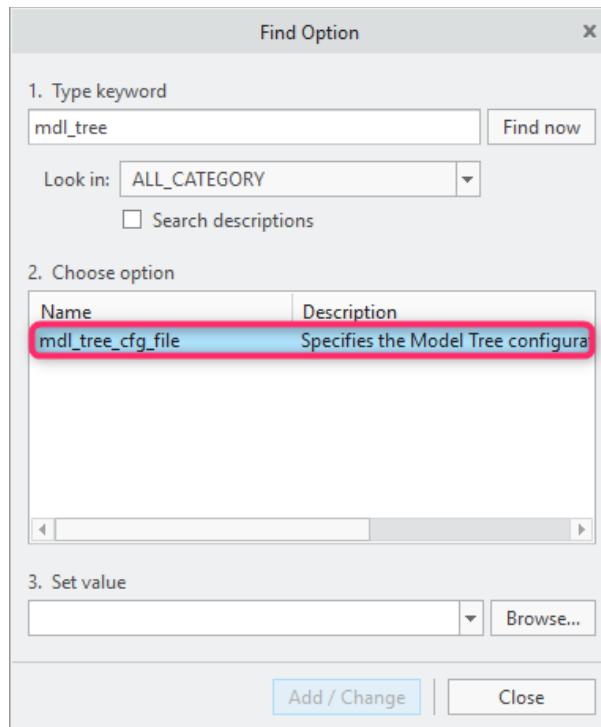
Select 

1. Type *mdl_tree* in the keyword search



Select

- The option will be automatically populated



Find Option

1. Type keyword

mdl_tree

Look in: ALL_CATEGORY

Search descriptions

2. Choose option

Name	Description
mdl_tree_cfg_file	Specifies the Model Tree configura

3. Set value

- Select *Set value* and browse to the directory you saved the *tree.cfg* file to

Highlight the file and select

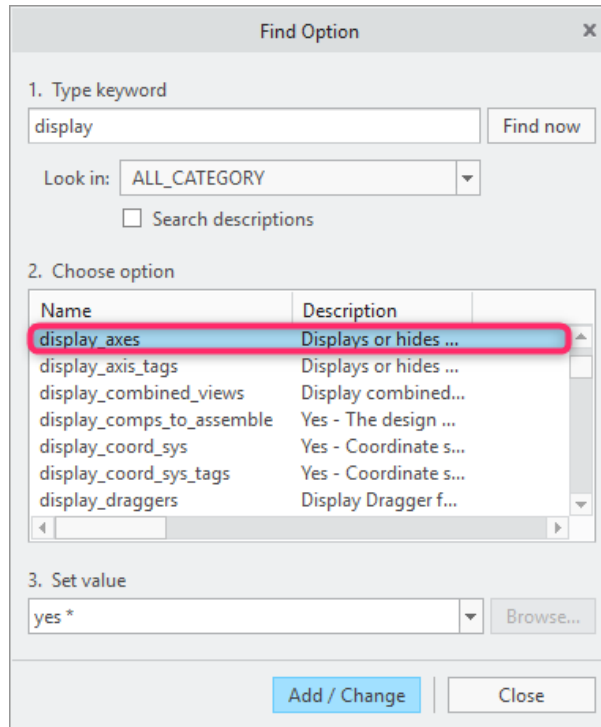
Select

To change the display configurations, repeat the process:

- Type *display* in the keyword search

Select

2. Select *display_axis* from the options



Find Option

1. Type keyword
 display Find now

Look in: ALL_CATEGORY
 Search descriptions

2. Choose option

Name	Description
display_axes	Displays or hides ...
display_axis_tags	Displays or hides ...
display_combined_views	Display combined...
display_comps_to_assemble	Yes - The design ...
display_coord_sys	Yes - Coordinate s...
display_coord_sys_tags	Yes - Coordinate s...
display_draggers	Display Dragger f...

3. Set value
 yes * Browse...

Add / Change Close

3. Select *no*

Select Add / Change

Repeat the process:

2. Select *display_coord_sys* from the options

3. Set value
 Select *no*

Select Add / Change

3. Select *display_planes* from the options

3. Set value
 Select *no*

Select Add / Change

Repeat the process:

2. Select *display_points* from the options

3. Set value

Select

Select

3. Select *display* from the options

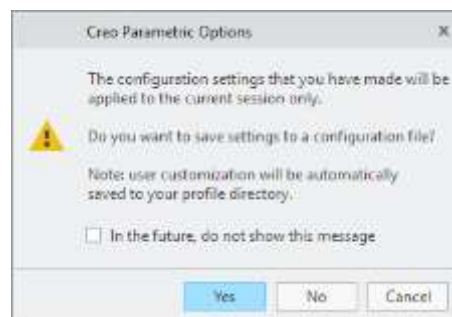
3. Set value

Select

Select

Select

Select



Select

Browse to the C:\Program Files\PTC\Creo #.#\M###\Common Files\text directory

Highlight *config.pro* and select