config.pro - Creo General Configuration File

Outline of Parameter Modifications

PARAMETER	NEW VALUE	DEFAULT VALUE
drawing_setup_file	170dwgconfig.dtl	
This parameter points Creo to a drawing setup file. The drawing setup file contains parameters that specify line widths, arrow sizes, and tolerance options.		
tolerance_standard	iso	ansi
Defines the default tolerance standard.		
tolerance_class	medium	
Defines the default tolerance class for ISO dimensioning.		
tol_display	no	no
Controls the default display of tolerances in the modeling environment. Tolerance display in drawings is controlled by the drawing setup file.		
orientation	isometric	trimetric
Defines the orientation to be used for a part's "standard orientation."		
default_dec_places	2	2
Defines the default number of decimal places for new dimensions.		
mdl_tree_cfg_file	170modeltree.cfg	
This parameter points Creo to a model tree configuration file. The 170modeltree.cfg file specifies a model tree where all model features are visible in the tree.		
sketcher_starts_in_2d	yes	no
Throughout the semester, we have had to press "Sketch View" to orient the sketcher to the proper 2D sketching plane. The sketcher will automatically enter "Sketch View" with this parameter set.		
disp_regen_success_msg	yes	no
Instructs Creo to display a "regeneration successful" message in the message log on each successful regeneration.		
display_point_tags	yes	no
Sets the default display of datum point tags.		
display_plane_tags	yes	no
Sets the default display of datum plane tags.		
display_axis_tags	yes	no
Sets the default display of axis tags.		
comp_assemble_start	constrain_in_window	
Sets the initial assembly behavior when assembling a new component. This parameter value instructs Creo to open a new, smaller window for viewing the component while it is being assembled.		
spin_with_part_entities	yes	
Instructs Creo to display datum features while a part is spun. Enabling this parameter requires more visual processing than the default option.		
spin_with_silhouettes	yes	no
Instructs Creo to display silhouette part outlines while spinning a part. Enabling this parameter requires more visual processing than the default option.		