

# **PTC Creo<sup>®</sup> 3.0 Parametric Configuration Options**

**Datecode F000**

**PTC Inc.**

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## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Application Programming Interfaces	add_java_class_path	Prepends the option value on the java environment variable, CLASSPATH, used to find classes in J-Link programs. To specify multiple search paths in the same line, delimit them using ':' on UNIX, and ';' on Windows NT.		
Application Programming Interfaces	enable_protk_xml_schema	Enables XML-Schema validation for Pro/TK feature XMLs	yes, no	no
Application Programming Interfaces	jlink_java2	Option to force use of Java-2 JRE command line ("java" instead of "jre")	off, on	on
Application Programming Interfaces	prodevdat	Specify the full path to an additional Toolkit registry file.		
Application Programming Interfaces	protkdat	Specify the full path to an additional Toolkit registry file.		
Application Programming Interfaces	toolkit_registry_file	Specify the full path to an additional Toolkit registry file.		
Application Programming Interfaces	web_disable_js_command	Enter a specific javascript command to be disabled		
Application Programming Interfaces	web_link_file_read	Sets Pro/Web.Link permission to read data from the local file system.	yes, no	no
Application Programming Interfaces	web_link_file_write	Sets Pro/Web.Link permission to write data to the local file system.	yes, no	no
Application Programming Interfaces	web_link_proe_read	Sets Pro/Web.Link permission to read data from a Creo Parametric session.	yes, no	no
Application Programming Interfaces	web_link_proe_write	Sets Pro/Web.Link permission to modify data in a Creo Parametric session.	yes, no	no
Application Programming Interfaces	web_link_security_autoprompt	Yes - The Security dialog box is displayed when a Pro/Web.Link function requires security access beyond the current settings.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Assembly	add_offset_to_constraint_comp	Adds an offset dimensions to constraints that were created without an offset during the dragging of a component.	yes, no	no
Assembly	advanced_intersection	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	no
Assembly	allow_freeze_failed_assy_comp	Allow freezing failed assembly components.	yes, no	no
Assembly	allow_package_children	Sets what type of children packaged components can have. ALL - Features and components can be a children of packaged component. FEAT - Only features can be children of packaged components. NONE - Packaged components cannot have children.	all, feat, none	all
Assembly	allow_redo_intersections	Yes - Displays the ReIntersect command in the ASSY FEAT menu.	yes, no	no
Assembly	allow_save_as_instance	Allows the option to save a copy of the active model as a family table instance.	yes, no	no
Assembly	allow_save_failed_model	Yes - Failed models can be saved. No - Failed models cannot be saved. Prompt - Let the user decide whether failed models can be saved.	yes, no, prompt	prompt
Assembly	allow_update_part_simp_rep_data	Allow updating part Simplified Rep data in session automatically when switching from one Simp Rep to another. Yes - Allow updating part simplified rep data. No - Do not update part simplified rep data. Prompt - Prompt the user if the current simplified rep should be updated.	yes, no, prompt	yes
Assembly	annot_elem_params_with_units	Some annotation element virtual parameters which were previous passed unitless will not be passed as attributes with units.		1
Assembly	auto_assembly_with_notebooks	Yes - Enables automatic assembly. No - Disables automatic assembly.	yes, no	yes
Assembly	auto_constr_always_use_offset	Control whether auto constraint should create offsets. Yes - Auto constraint always creates offsets. No - Auto constraint snaps align or mate if surfaces are within tolerance. Never - Auto constraint never create offsets.	yes, no, never	no
Assembly	auto_evaluate_simp_prep_rules	If the option is "yes" newly created rep would be updated each time on retrieve/regeneration.	yes, no	no
Assembly	auto_place_max_number	Set the default value for the maximum number of matches allowed.		5
Assembly	auto_update_intersected_models	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	yes
Assembly	autoplace_single_comp	Assembling components with an interface will automatically place the component at the first position satisfying the interface definitions.	yes, no	yes

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Category	Name	Description	Values	Default Value
Assembly	bump_revnum_on_retr_regen	Determines whether or not revision number is increased for generic models that regenerate and change during assembly retrieval. Only applies if new_asm_regen_revnums is yes (otherwise, there will be no revision number bumping).	yes, no	yes
Assembly	can_snap_to_missing_ref	Controls whether to enable snapping to missing references while dragging components for placement. Yes - Snapping is enabled while dragging. No - Snapping is disabled while dragging.	yes, no	yes
Assembly	check_interface_criteria	Check for interfaces when placing a component.	yes, no	no
Assembly	check_interference_of_matches	Show only matches that do not interfere with other components.	yes, no	yes
Assembly	check_same_assy_parts	Determines whether to permit different names for different part occurrences, or not in mirror subassembly UI.	yes, no	yes
Assembly	chooser_size_filter_default	Sets the size filter default value as a percentage of the overall assembly size		
Assembly	comp_angle_offset_eps	Specifies the angle epsilon such that if the desired surfaces are equal or greater than the epsilon (in degrees), than an angle offset constraint will be created.		10
Assembly	comp_assemble_start	Sets the initial assembly placement behavior when assembling a new component. Default - Default initial position. Package - Define initial position using the Move option. Constraint_in_Window - Use secondary window. Move_then_Place - Define initial position using the Move option and then define placement constraints.	package, constrain_in_window, default, move_then_place	
Assembly	comp_placement_assumptions	Use placement assumptions when placing components.	yes, no	yes
Assembly	comp_retr_angular_increment	Sets the accuracy for the selection of external components based on graphic computation.		1
Assembly	comp_rollback_on_redef	No - Assembly is not rolled back when the user redefines a component.	yes, no	yes
Assembly	copy_geom_update_pre_2000i_dep	Yes - Flags independent copy geom features in a pre-2000i model as modified when retrieved into Creo Parametric. Save the model immediately to update the model's copy geom dependency information.	yes, no	
Assembly	copy_geometry_method	Sets the default copy geometry reference(s) type	publish_geometry, references	publish_geometry
Assembly	create_temp_interfaces	Allow the automatic creation of interfaces based on previous assembly instructions.	yes, no	no
Assembly	dim_inactive_components	Display inactive assembly components with stippled transparency and default color (grey). Default value is shade_only.	never, always, shaded_only	shaded_only

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Category	Name	Description	Values	Default Value
Assembly	disp_regen_success_msg	Issue successful regeneration messages for assembly models.	yes, no	no
Assembly	enable_advance_collision	Enables advance collision settings usage.	yes, no	no
Assembly	enable_assembly_accuracy	No - Disables accuracy modification for assembly objects.	yes, no	yes
Assembly	enable_implied_joints	Allow underconstrained components as mechanism connections.	yes, no	yes
Assembly	erv_show_external_only	Yes - Global Reference Viewer will show objects with external references only; No - all the objects will be visible	yes, no	no
Assembly	fail_ref_copy_when_missing_orig	Yes - When using a copied geom feature or a backed-up external reference with the Design Manager, and the reference is missing in the original, then the copied geom feature fails. No - The feature freezes and does not update.	yes, no	yes
Assembly	fix_refs_to_intersections	Sets remnant removal ability for the RelIntersect command in the ASSY FEAT menu. Yes - You can remove remnants. No - You cannot remove remnants.	yes, no	no
Assembly	force_upd_assem_mp_in_simp_rep	Update mass properties parameters even when some of the components of the assembly are not in master rep.	yes, no, use_stored	use_stored
Assembly	grv_default_view	Sets the default view the reference view starts with - Dependencies or References.	references, dependencies	references
Assembly	ignore_non_displayed_items	Ignores nondisplayed items when searching the component interface during component placement.	yes, no	no
Assembly	include_sub_model_interfaces	Includes sub model interfaces as possible references for interface to interface component placement.	yes, no	yes
Assembly	interface_criteria_mismatch	FAIL_PLACEMENT - Fail regeneration upon component interface criteria mismatch. WARNING_ONLY - A warning message is given during regeneration for the criteria mismatch only without failure. IGNORE - Interface criteria mismatch ignored during regeneration.	ignore, warning_only, fail_placement	warning_only
Assembly	modify_offset_during_comp_drag	Enables the modification of the value of offset dimensions during component dragging.	yes, no	no
Assembly	mp_dens_ft_column_mtrl_assigned	Yes - Allows adding an MP_DENSITY column in FT when a material is assigned to a model. When set may cause the same material to have different values for the Density parameter in different instances. Does not affect legacy models.	yes, no	no
Assembly	mtrl_convert_units	If set to "Yes" Creo Parametric will automatically convert units in materials from a library to the model system of units.	yes, no	yes
Assembly	new_wf5_simp_rep_ui	Yes - Use Component Chooser base Simp Rep UI, No - Use old Simp Rep definition UI.	yes, no	yes

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Category	Name	Description	Values	Default Value
Assembly	package_constraints	Action to take for packaged components in Assembly mode. Update - Updates components if assembly changes. Freeze - Does not move components if assembly changes. Disallow - Does not allow packaged components in assembly (must be fully constrained).	disallow, freeze, update	update
Assembly	package_ref_alert	Control whether alert dialog is given when packaged component is referenced. Yes - Allow alert when a packaged component is referenced. No - No Alert is given when a packaged component is referenced.	yes, no	no
Assembly	preserve_comp_color_in_preview	Yes - During component placement, component being assembled will retain its normal colors. No - During component placement, component being assembled is shown in preview color.	yes, no	no
Assembly	regen_notebook_w_assem	Specifies whether or not the notebook will be automatically regenerated upon assembly regeneration.	yes, no	yes
Assembly	replace_comp_name_method	Sets the naming convention used when replacing components.	transfer, remove, preserve	transfer
Assembly	replace_history_rule_order	Set the default order of the history evaluation rule in the replace operation. Disable - the evaluation rule will be disabled by default. Number - set the order of the rule relative to other evaluation rules.	1, 3, 2, disable, 4, 5	3
Assembly	replace_interface_rule_order	Set the default order of the interface evaluation rule in the replace operation. Disable - the evaluation rule will be disabled by default. Number - set the order of the rule relative to other evaluation rules.	1, 3, 2, disable, 4, 5	1
Assembly	replace_sameid_rule_order	Set the default order of the same id evaluation rule in the replace operation. Disable - the evaluation rule will be disabled by default. Number - set the order of the rule relative to other evaluation rules.	1, 3, 2, disable, 4, 5	5
Assembly	replace_samename_rule_order	Set the default order of the same name evaluation rule in the replace operation. Disable - the evaluation rule will be disabled by default. Number - set the order of the rule relative to other evaluation rules.	1, 3, 2, disable, 4, 5	2
Assembly	replace_sameparam_rule_order	Set the default order of the same parameters evaluation rule in the replace operation. Disable - the evaluation rule will be disabled by default. Number - set the order of the rule relative to other evaluation rules.	1, 3, 2, disable, 4, 5	4
Assembly	replace_unrelated_automatic	Availability of automatic reference pairing in unrelated replace. No - Automatic reference pairing is not available. Yes - Automatic reference pairing is available.	yes, no	no
Assembly	retrieve_instance_dependencies	Determines whether to retrieve the dependencies of the generic when an instance is retrieved.	instance_deps_only, instance_and_generic_deps, instance_req_generic	instance_req_generic

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Category	Name	Description	Values	Default Value
Assembly	rv_current_object	Will control default option when setting a model or component as the current object.	model, component, component + model	component + model
Assembly	rv_search_scope	Sets the search scope for children when entering the reference viewer. Session - Search all objects in session. Window - Search objects in the active window only.	session, window	session
Assembly	search_area_for_comp_interfaces	Defines the search area for the component interface as a percentage of the component size during component placement.		100
Assembly	search_not_retrieved_models	Expands search to include models that have not been retrieved into session. If Yes, models will be brought into session possibly causing performance issues.	yes, no	yes
Assembly	sel_insts_on_comp_retrieval	Yes - If the components used in the assembly instances are themselves generics, the systems asks if you want to choose an instance as each component is retrieved. No - The generic model of the component is retrieved automatically.	yes, no	no
Assembly	show_interference_in_sections	Yes - Interference will be shown automatically, No - interference will not be shown automatically.	yes, no	no
Assembly	shrinkwrap_alert	Displays the Shrinkwrap alert the first time the quality level is increased.	yes, no	yes
Assembly	simple_search_consider_select	Defines whether simple search executes query only inside selected models (if some models selected) or always in scope of entire top-level model.	yes, no	no
Assembly	simprep_default_model_statuses	Determines the default model status while defining a new simplified rep.	master, geometry, exclude, graphics, light graphics rep	exclude
Assembly	simprep_ondemand_selection	Determines the rep to be retrieved when selecting a reference. Automatic Retrieves the minimum rep required to perform the operation.	automatic, master, disable	automatic
Assembly	simprep_ondemand_settings	Determines on-demand retrieval behavior in simplified representation views. Prompt - Ask for confirmation before retrieving on-demand. Never_prompt - Retrieve models on-demand automatically with no confirmation. Disabled - on demand is disabled.	prompt, disable, never_prompt	never_prompt
Assembly	skeleton_model_default_color	Specifies the color Creo Parametric uses to display new skeleton models. The three decimal values ranging from 0 through 100 specify (in order) the percentages of red, green, and blue in the resulting color. For example, 0 0 49 specifies medium blue.		0.000000 75.000000 100.000000
Assembly	snap_to_constraints_during_drag	Enables the dynamic snapping to constraints during the dragging of components.	yes, no	yes

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Category	Name	Description	Values	Default Value
Assembly	update_copy_geom_location	Yes-Copy Geometry will try to update location of copied geometry based on changes to component placements even if some components are excluded or in graphics rep.	yes, no	yes
Assembly	update_rep_refs	"Yes" updates top assembly simplified reps for replacements during regeneration and upon replacement.	yes, no	yes
Assembly	use_3d_thumbnail_in_lwg_rep	Control the usage of 3D thumbnails when retrieving in light weight graphic representation	yes, no	yes
Assembly	use_active_model_in_transform	Transform analysis default behavior: use active model CSYS and units or not.	yes, no	no
Assembly Process	curr_proc_comp_def_color	Specifies the default color used to display the current component in a process assembly.		0.000000 0.000000 0.000000
Assembly Process	curr_proc_comp_def_font	Specifies the default font used on the current component in a process assembly.		
Assembly Process	display_comps_to_assemble	Yes - The design model goes into memory and displays. Pick process components from the design model or the Model Tree. No - Only the Model Tree displays. As you pick process components from the Model Tree, they go into memory.	yes, no	yes
Assembly Process	offset_line_def_color	Specifies the color that offset lines are displayed in drawings.		0.000000 0.000000 0.000000
Assembly Process	offset_line_def_font	Specifies the font in which offset lines are displayed in drawings.		
Assembly Process	prev_proc_comp_def_color	Specify the default color for components added to a process assembly in a previous step.		0.000000 0.000000 0.000000
Assembly Process	prev_proc_comp_def_font	Specify the default font for previously added components in a process assembly.		
Casting & Mold Design	allow_shrink_dim_before	Allow calculation before relations.	yes, no	no
Casting & Mold Design	default_mold_base_vendor	Default Value for Mold Base vendor.	futaba_mm, dme, hasco, dme_mm, hasco_mm	
Casting & Mold Design	default_shrink_formula	Default option of shrinkage formula: 1/(1-S) - ASME standard or 1+S (simple).	asme, simple	simple
Casting & Mold Design	mold_layout_origin_name	Sets a specified coordinate system as the default for the cavity layout origin.		
Casting & Mold Design	mold_vol_surf_no_auto_rollback	Do not perform automatic rollback while modifying mold volumes or parting surfaces.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Casting & Mold Design	pro_catalog_dir	Sets the path to the catalog directory containing the catalog menu and the names of other catalog files, such as ejector pins. For example: pro_catalog_dir proe_loadpoint/apps_data/mold_data/catalog. Use full path name to avoid problems.		
Casting & Mold Design	pro_cav_lay_rule_dir	Sets the default directory for cavity layout rules. Use the full path name to avoid problems.		
Casting & Mold Design	shrinkage_value_display	Determines how dimensions are displayed when shrinkage is applied to a model with Pro/MOLDESIGN. Percent_shrink - Dimensions appear with the shrinkage percent.	final_value, percent_shrink	percent_shrink
Colors	color	Turns the colors on or off. Turning colors off displays the model wireframe in white.	yes, no	yes
Colors	color_ramp_size	Specifies the number of shades in a color ramp. Applicable to multi-color, shaded model displays of simulation / analysis results. System graphics must support 256 colors and color maps that compress.		-1
Colors	color_resolution	Specifies minimum allowable deviation between user-defined colors. Colors having RGB values within tolerance of existing colors cannot be created. Decreasing this setting allows more colors that are very similar in RGB value to be defined.		0.1
Colors	global_appearance_file	Material Global Appearance file path.		
Colors	mat_assign_appearance	Control whether default appearance in material definition is automatically assigned to a part.	yes, no	yes
Colors	number_user_colors	Specifies the maximum number of different wireframe colors that can display in the graphics area(s) at any given moment.		-1
Colors	pro_colormap_path	Specifies the directory path for a color map (.map) file to be loaded from disk. Use the full path name to avoid problems.		
Colors	reflection_analysis_display	Black and White displays reflection as black and white stripes. Surface color - uses the surface color to represent reflection stripes.	black and white, surface color	black and white
Colors	suppress_appearance_message	Suppress the Lightworks appearance conversion notification dialog.	yes, no	no
Colors	system_background_color	Defines default graphics area background color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_colors_file	Specifies the system color file. Use the full path name to avoid problems.		

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Category	Name	Description	Values	Default Value
Colors	system_curves_color	Defines default curve color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_dimmed_menu_color	Defines default secondary highlight color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_edge_high_color	Defines default edge highlight color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_geometry_color	Defines default color of solid wireframe entities. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_hidden_color	Defines default color of hidden, wireframe entities. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_highlight_color	Defines default primary highlight color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_letter_color	Defines default color of datum tags. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_section_color	Defines default section color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	system_sheetmetal_color	Defines default color of Sheetmetal parts. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000

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Category	Name	Description	Values	Default Value
Colors	system_volume_color	Defines default color of Manufacturing volumes. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.		0.000000 0.000000 0.000000
Colors	use_part_color_for_hidden_lines	Determines how to color the hidden lines of parts that have user defined colors. "YES" - Use the dimmed part color for hidden lines. SYSTEM_HIDDEN_COLOR is ignored if it has been specified.	yes, no	no
Creo Simulate	custom_rainbow	Used to specify fringe colors in Pro/FEM-POST results. Specify blue, black, cyan, magenta, green, yellow, red, and white. Enter colors as a dash-delimited string (lowest to highest value). The default is blue-cyan-green-yellow-red-magenta-white.		
Creo Simulate	sim_hyperelastic_material_fit	Sets the default method for calculating coefficients for hyperelastic materials. If set to "normalized", normalized stresses will be used. If set to "non-normalized", non-normalized stresses will be used.	normalized, non-normalized	normalized
Data Exchange	acis_export_params	For ACIS interface allows to export object, feature and entity level parameters.	yes, no	no
Data Exchange	acis_export_units	Allows selection of model units for ACIS export.	micron, mm, default, in, ft, m, cm	default
Data Exchange	afx_enabled	Loads AFX application.	yes, no	yes
Data Exchange	allow_flat_to_screen_note	Allow creation of flat to screen notes when option is set to true	yes, no	no
Data Exchange	atb_auto_check_on_activate	Sets state of ATB 'Auto Check Status On Activate' toggle.	off, on	off
Data Exchange	atb_auto_check_on_retrieve	Sets state of ATB 'Auto Check Status On Retrieve' toggle.	off, on	on
Data Exchange	atb_auto_check_on_update	Sets state of ATB 'Auto Check Status On Update' toggle.	off, on	off
Data Exchange	atb_ident_cadds_files	Allows to identify CADDs files in the file open browser.	yes, no	no
Data Exchange	atb_prod_asm_upd_by_comp_ids	Allows to update a Pro/DESKTOP assembly based on assembly component ids, if set to YES. Otherwise, the update will be based on component names.	yes, no	yes
Data Exchange	atb_show_foreign_name	Displays foreign source model name in the model tree for imported models.	yes, no	yes
Data Exchange	atb_show_log	Sets state of ATB 'Show Log' toggle.	off, on	on
Data Exchange	auto_associate_dimensions	If set to yes, Creo Parametric will attempt to take imported dimensions that are not associative and link them to geometric entities so that they act as associative.	yes, no	no

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Category	Name	Description	Values	Default Value
Data Exchange	board_bend_notes_dflt_display	Define the default state of bend note display (board mode)	yes, no	yes
Data Exchange	board_bend_notes_direction_down	Defines the down direction symbol for board bend notes.		default
Data Exchange	board_bend_notes_direction_up	Defines the up direction symbol for board bend notes.		
Data Exchange	board_bend_notes_order	Defines the fields order for board bend notes.		&type&direction&angle
Data Exchange	board_bend_notes_type_formed	Defines the formed symbol for board bend notes.		default
Data Exchange	board_bend_notes_type_rolled	Defines the rolled symbol for board bend notes.		default
Data Exchange	board_bend_table_directory	Will be used to set the bend allowance table only. Sets the default directory of the board bend allowance table files. If not set, the system supplied bend tables are used. Use the full name and path to avoid problems.		
Data Exchange	board_design_rules_directory	Will be used to set the design rules file folder only. Will be used by the design rules manager. Sets the default directory of the board design rules files. Use the full name and path to avoid problems.		
Data Exchange	board_drive_bend_by_parameters	Board bend radius and bend angle values and bend dimension location will be driven by feature-level relations to board parameters.	yes, no	no
Data Exchange	board_drive_tools_by_parameters	Drives the board options and values by feature-level relations to board parameters.	yes, no	yes
Data Exchange	board_flange_shape_sketches_dir	Sets path to directory that contains the Flange shapes.		
Data Exchange	board_flat_shape_sketches_dir	Sets path to directory that contains the Flat shapes.		
Data Exchange	board_parameters_directory	Specifies the directory to save/retrieve board parameters files. Use full name and path to avoid problems.		
Data Exchange	cadds_import_layer	Allows import of CADD5 layering.	yes, no	yes
Data Exchange	catia_out_to_existing_model	Append - If the selected CATIA model already exists, the new data is appended to the existing CATIA file. Overwrite - If the selected CATIA model already exists, the newly exported file overwrites the existing one.	append, overwrite	append
Data Exchange	cgm_use_pentable	No - Do not use a defined pen table for formatting of CGM export.	yes, no	no
Data Exchange	cri_grafting_enable	Activates "Graft features" button under File menu. This allows user to graft features created in CRI model to active model.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	dazix_default_placement_unit	Specifies the units to be used for data imported in Dazix files.	micron, mm, thou	
Data Exchange	dazix_export_mounthole	Yes - Causes the MOUNTHOLE section of a Dazix file to be processed as a mount hole. No - Causes the MOUNTHOLE section to be processed as a cut.	yes, no	no
Data Exchange	dazix_z_translation	Yes - Passes the objects in the .edn files through z translation.	yes, no	yes
Data Exchange	direct_vrml	Yes - Enables direct VRML conversion.	yes, no	no
Data Exchange	dwg_export_format	Provides the ability to choose DWG file version when exporting from Creo Parametric drawing.	14, 2000, 2004, 2007, 2010, 2013	2013
Data Exchange	dxblock_to_pro_symbol	Specifies whether or not DXF BLOCKS to be imported as Creo Parametric symbols. Yes - Imports BLOCKS as symbol definitions, No - Imports BLOCKS as separate entities.	yes, no	no
Data Exchange	dx_export_format	Provides the ability to choose DXF file version when exporting from Creo Parametric drawing.	12, 13, 14, 2000, 2004, 2007, 2010, 2013	2013
Data Exchange	dx_in_faceted_brep	Default, faceted boundary representation is allowed in import.	yes, no	yes
Data Exchange	dx_out_comments	YES - comments (999 group codes) will be created in DXF file exported from Creo Parametric. NO - DXF file will be created without comment lines.	yes, no	yes
Data Exchange	dx_out_drawing_scale	Specify whether or not to export drawing scale to the DXF or DWG file. Yes - Includes a scale factor DIMLFAC in the exported DXF/DWG. No - Exports without using a scale for the drawing. Set to Yes when exporting to read into AutoCAD.	yes, no	no
Data Exchange	dx_out_scale_views	Specify whether or not to scale drawing views when exporting to the DXF or DWG file. Yes - rescales the entire drawing in such a way that the main view scale becomes 1:1. No - exports without rescaling.	yes, no	no
Data Exchange	dx_out_sep_dim_w_breaks	YES - Breaks dimensions with breaks on witness lines to separate entities and preserves the original picture. No - Exports such dimensions as DXF DIMENSION entities and removes the breaks on witness lines.	yes, no	no
Data Exchange	dxio_in_sjis	Determines if the SJIS is supported when importing/exporting DXF files in Japanese	yes, no	yes
Data Exchange	ecad_area_default_import	Defines how imported ECAD areas are treated. Cosm_area - Treats imported ECAD areas as cosmetic area features. 3d_volume - Imports ECAD areas with Z-heights as a 3D enclosed quilt.	3d_volume, cosm_area	cosm_area

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	ecad_board_csys_def_name	Specifies the default coordinate system name added to an ECAD board being imported. If you do not set this variable, the system prompts you for a name.		
Data Exchange	ecad_comp_csys_def_name	Specifies the default coordinate system name added to an ECAD component being imported. If you do not set this variable, the system prompts you for a name.		
Data Exchange	ecad_comp_layer_map	Allow layer mapping for ECAD component's import into Assembly.	yes, no	yes
Data Exchange	ecad_comp_naming_convention	ECAD_NAME / ECAD_ALT_NAME - Uses the components package name or part number as the part name. ECAD_NAME_ECAD_ALT_NAME - Concatenates the component's package name and part number as the part name.	ecad_name, ecad_name_ecad_alt_name, ecad_alt_name	ecad_name
Data Exchange	ecad_comp_xsec_def_name	Will allow pre-setting default X-section to be used for component outline output to *.eda format.		
Data Exchange	ecad_create_hint_add	Assists in creation of an ecad_hint.map file. Yes - Automatically renames components, if necessary, each time library of component outlines is imported to Creo Parametric. This does not create an ecad_hint.map file. It controls if ecad_hint.add is created.	yes, no	yes
Data Exchange	ecad_default_comp_height	Sets default value and units for an ECAD component being imported. Units can be: inch, mil (1E-3 inches), thou (1E-6 inches), cm, mm, micron (1E-6 meters), dsu (1E-8 meters). If not set, the system uses the current component's units.		-1.000000 ecad_unit_not_set
Data Exchange	ecad_default_comp_place_status	Sets default component placement status for export (parameter setting will override the default).	unplaced, fixed, placed, mcad, ecad	placed
Data Exchange	ecad_edmd_out_version	Set the EDMD output version.	2.0, 1.2, 3.0	2
Data Exchange	ecad_exp_both_two_areas	Supports the export of ECAD areas with different "Above Board" and "Below Board" conditions. "yes" - Enables you to export both sided keep-in/keep-out ECAD areas as two individual areas (top and bottom).	yes, no	no
Data Exchange	ecad_export_cuts_as_holes	Exports Creo Parametric cuts as holes to ECAD systems.	yes, no	yes
Data Exchange	ecad_export_holes_as_cuts	Exports Creo Parametric holes as cuts to ECAD systems.	yes, no	no
Data Exchange	ecad_import_holes_as_features	Import sections specified as DRILLED_HOLE as through-all holes. Boards created with Creo Parametric drilled holes export with default value of NPTH for the ECAD_HOLE_TYPE parameter. Create this feature parameter if a value of PTH is needed (IDF 2.0/3.0).	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	ecad_import_relative_accuracy	Defines the relative accuracy used in ECAD import.		0.0012
Data Exchange	ecad_load_filtered_holes_to_ui	Controls whether "ECAD import holes configs" (ecad_pin_hole_import & ecad_via_hole_import) will remove holes completely from the import (default) or just pre-set the holes filter UI condition.	yes, no	no
Data Exchange	ecad_mapping_file	Specify the ecad_hint.map file that will be used for ECAD operations.		
Data Exchange	ecad_missing_component_status	Setup default status for missing components in "Investigate Placement" Dialog.	keep_missing, delete_missing	keep_missing
Data Exchange	ecad_mtg_hole_import	Setup the default for the import of ECAD Holes type MTG (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
Data Exchange	ecad_other_outl_csys_def_name	Specifies the default coordinate system name for the .OTHER_OUTLINE section of an IDF 2.0 ECAD component being imported. If you do not set this variable, the system uses ECAD DEFAULT as coordinate system name.		
Data Exchange	ecad_outline_ents_enhanced_exp	Enhanced export of special entities to ECAD formats. Yes - Attempt to approximate arcs and lines. No - Use part accuracy to approximate lines.	yes, no	yes
Data Exchange	ecad_outline_holes_exp_method	Select how holes that are placed on the board outline be exported.	default, arcs	default
Data Exchange	ecad_panel_csys_def_name	Specifies the default coordinate system name added to an ECAD panel being imported. If you do not set this variable, the system prompts you for a name.		
Data Exchange	ecad_pin_hole_import	Set up the default for the import of ECAD Holes from type PIN (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	no
Data Exchange	ecad_reject_strategy	Configures the way the changes are rejected.	ui_based, as_original, as_base	ui_based
Data Exchange	ecad_set_intercomm_compatible	Yes - Set both EDA format version and output method to support InterComm product, yes or no.	yes, no	no
Data Exchange	ecad_tool_hole_import	Setup the default for the import of ECAD Holes type TOOL (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
Data Exchange	ecad_via_hole_import	Set up the default for the import of ECAD Holes from type VIA (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
Data Exchange	ecadcollab_accept_color	ECAD Collaboration Accept color		0.0 255.0 0.0
Data Exchange	ecadcollab_auto_redef_areas	Automatically redefine ECAD Areas affected by collaboration changes	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	ecadcollab_auto_redef_comps	Automatically redefine components affected by collaboration changes	yes, no	yes
Data Exchange	ecadcollab_auto_redef_holes	Automatically redefine holes affected by collaboration changes	yes, no	yes
Data Exchange	ecadcollab_preview_after_color	ECAD Collaboration Preview After color		255.0 127.0 0.0
Data Exchange	ecadcollab_preview_before_color	ECAD Collaboration Preview Before color		0.0 76.0 255.0
Data Exchange	ecadcollab_preview_level	ECAD Collaboration preview Level	full, partial, light	partial
Data Exchange	ecadcollab_reject_color	ECAD Collaboration Reject color		255.0 127.0 127.0
Data Exchange	ecadcollab_scan_des_on_areas_chg	Check for affected objects resulting from ECAD Areas collaboration changes	yes, no	yes
Data Exchange	ecadcollab_scan_des_on_board_chg	Check for affected objects resulting from board collaboration changes	yes, no	yes
Data Exchange	ecadcollab_scan_des_on_components_chg	Check for affected objects resulting from components collaboration changes	yes, no	yes
Data Exchange	ecadcollab_scan_des_on_holes_chg	Check for affected objects resulting from holes collaboration changes	yes, no	yes
Data Exchange	ecadcollab_supp_failed_feats	Suppress of failed features on Object changes	yes, no	yes
Data Exchange	ecadcollab_suppress_cuts	Automatically suppress cut features in case of board collaboration changes	yes, no	yes
Data Exchange	enable_acis_export_extension	Enable ACIS export extension.	yes, no	no
Data Exchange	enable_cadra_export	Enables the CADRA command in the EXPORT menu. This enables the creation of a CADRA specific IGES file.	yes, no	no
Data Exchange	explode_iges_dimension_note	Controls how dimensions are treated when importing an IGES drawing file. Yes - Each IGES dimension explodes into an independent note with the dimension text and a dimension with an empty note. No - Dimensions are treated as before.	yes, no	no
Data Exchange	export_3d_force_default_naming	Specify whether the Creo Parametric model names should be used for STEP export. Yes - Use default naming. No - Add suffixes to model file names.	yes, no	no
Data Exchange	export_to_shipit	Enable Ship-it interface export.	yes, no	no
Data Exchange	extend_cvpath	Extends CVPATH search path for CADD5 data exchanging.		
Data Exchange	fix_autocad_iges_text_scale	Fixes scaling problems with AutoCAD releases earlier than Release 10. Yes - Repairs AutoCAD files appropriately.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	fix_catia_iges_sym_note	Corrects a problem caused by multiple rotations of a note in a CATIA IGES file. Yes - Rectifies any discrepancies.	yes, no	no
Data Exchange	fix_imported_set_view_orient	Sets viewing characteristics for imported CATIA SET files. Yes - Files can have translated views. No - Files cannot have translated views.	yes, no	no
Data Exchange	frt_enabled	Loads feature recognition application.	yes, no	no
Data Exchange	iges_clip_view_ent	Determines whether IGES entities are clipped with regards to IGES views. YES - Entities outside the view outline are clipped. NO - No clipping.	yes, no	yes
Data Exchange	iges_clip_view_note	Determines how IGES notes are clipped with regards to IGES views. No_clip - No clipping. Full_clip - Notes that are partially outside the view outline are clipped. Partial_clip - Notes completely outside the view outline are clipped.	no_clip, full_clip, partial_clip	no_clip
Data Exchange	iges_export_dwg_views	Yes - Exports drawing view information. No - Does not export drawing view information.	yes, no	no
Data Exchange	iges_in_dwg_color	Yes - Import RGB information to IGES files. The color definition entities in the IGES file are imported to the drawings as user-defined colors. No - The color-definition entities in the IGES files are ignored when importing to the drawing.	yes, no	no
Data Exchange	iges_in_dwg_line_font	Controls the import of user-defined line fonts. Yes - Give default names to user-defined line styles without names in order ("IGES_1," "IGES_2" etc.). No - Imports the user-defined line fonts as Solid line font.	yes, no	no
Data Exchange	iges_in_group_to_dwg_layer	Yes - Converts an IGES group to drawing layers. No - The IGES group is not converted.	yes, no	no
Data Exchange	iges_note_disp	Determines what to do when IGES note entities do not point to any views or the drawing entity. See the configuration option iges_zero_view_disp.	as_geometry, all_views, no_views, as_is	all_views
Data Exchange	iges_out_all_srfs_as	Default - Outputs all surfaces as appropriate IGES surfaces. 114 - This option applies only to representations of surface shape. Trimmed surface entities (type 144) are exported regardless of setting. 128 - Outputs all surfaces as IGES B-spline surfaces.	128, 114, default	default
Data Exchange	iges_out_assembly_default_mode	Set default for assembly export via IGES. Flat - Single level model. One_level - Assy structure with pointers to components. All_levels - Assy structure with all comps to IGES files. All_parts - Assy to IGES files with geom info of comps and assy feats.	flat, one_level, all_levels, all_parts	flat
Data Exchange	iges_out_catia_gdt_width	Yes - Enables the desired width of a gtol symbol to be exported to CATIA.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	iges_out_catia_notes	Yes - Breaks a large note (greater than 70 strings) down into smaller notes, each producing an IGES entity.	yes, no	no
Data Exchange	iges_out_dwg_color	Yes - Enables the export of RGB information to IGES files. The user-defined colors in the drawing are exported to an IGES file as color definition entities. No - Ignores the user-defined colors in the drawing when exporting IGES file.	yes, no	no
Data Exchange	iges_out_dwg_line_font	Controls the export of user-defined line font through IGES. No - Exports all geometry as solid font.	yes, no	no
Data Exchange	iges_out_ent_as_bspline	True - Exports all geometry entities, other than lines or arcs, as third-degree B-splines. False - Does not export entities as third-degree B-splines.	true, false	FALSE
Data Exchange	iges_out_jamais_compliant	Yes - Specifies IGES output with special JAMA-IS subset specification, compliant with the version 1.02 of JAMA-IS (Japan Automobile Manufacturers Association IGES Subset Specification). No - Specifies normal IGES.	yes, no	no
Data Exchange	iges_out_mil_d_28000	Yes - Specifies IGES output that uses the MIL-D-28000 entity subset.	yes, no	no
Data Exchange	iges_out_spl_crvs_as_126	Converts all part geometry spline curves to IGES entity 126 (B-spline) when creating an IGES file.	yes, no	yes
Data Exchange	iges_out_spl_srfs_as_128	Converts all part geometry spline surfaces to IGES entity 128 (B-spline) when creating an IGES file.	yes, no	yes
Data Exchange	iges_out_start_note	Yes - Exports the text specified by system_iges_header_file and user_iges_header_file as a note on the drawing. To control note placement, use the configuration file option put_iges_drawing_entity.	yes, no	no
Data Exchange	iges_out_symbol_entity	Exports a drawing symbol as an IGES general symbol entity, IGES type number 228 (yes), or as its component entities: notes and lines.	yes, no	yes
Data Exchange	iges_out_trim_curve_deviation	Sets the maximum value for the distance between an XYZ trimming curve (boundary) and the underlying surface of a trimmed surface		-1
Data Exchange	iges_out_trim_xyz	Controls whether or not XYZ data is output in addition to UV data for trimmed surfaces.	yes, no	yes
Data Exchange	iges_out_trm_srfs_as_143	Yes - Exports all surfaces to IGES entities 141 and 143. This overrides option iges_out_trm_xyz.	yes, no	no
Data Exchange	iges_zero_view_disp	When IGES geometry does not point to any views or the drawing entity. All_views - Creates a copy of the entity for each view. No_views - Does not create the entity at all. As_is - Creates the entity once, using only its own transformation information.	all_views, no_views, as_is	all_views

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	initial_board_bend_y_factor	Specifies a value used to determine the neutral bend line for a board part. This value is always used for non-cylindrical bends. It is used for regular bends only when no bend table is specified		0.5
Data Exchange	interface_quality	Sets amount of checking for overlapping lines and collecting lines of same pen color before exporting plot or 2-D file. 0 - No check or collection. 1 - No check, but collection. 2 - Partial check and collection. 3 - Complete check and collection.		3
Data Exchange	intf_cadds_import_make_solid	Allows the user to solidify all closed quilts during import CADD5 model.	yes, no	no
Data Exchange	intf_cadds_version	Allows the user to switch between the default and alternate converter version for interface with CADD5	14, 15	14
Data Exchange	intf_generate_log_file	Specifies not to generate or generate short or long log files for data exchange import operations.	no, long, short	short
Data Exchange	intf_in_dwg_pnt_ent	Yes - Converts an IGES or DXF point entity to a drawing point.	yes, no	no
Data Exchange	intf_in_dwg_view	Controls if associativity in an imported IGES view is preserved. 3D_VIEWS - tries to create 3D views if 3D model exists in the file. 2D_VIEWS - Imported IGES views work as 2D. No - Imported IGES views are exploded and become unrelated.	no, 2d_views, 3d_views	2d_views
Data Exchange	intf_in_extract_profiles	Default, no profiles in a data exchange file will be extracted for use.	all, none, comp	none
Data Exchange	intf_in_granite_direct_enable	Allows the user to switch back to the old 'Import Feature' method of reading Desktop .des files and Granite .g files into Creo Parametric. By default, Creo Parametric directly opens these models.	yes, no	yes
Data Exchange	intf_in_layer_asm_dialog	Yes - Presents a dialog box to control the import of both layers and assemblies. You can choose from the list of layers and/or solids available in the file to select one or more for import.	yes, no	no
Data Exchange	intf_in_profile_default	Specifies the name of the existing import profile to use by default.		
Data Exchange	intf_out_as_bezier	Exports all B-splines as Bezier surfaces.	yes, no	no
Data Exchange	intf_out_asm_mapped_item	Export assembly components and instances using MAPPED ITEM entity.	yes, no	no
Data Exchange	intf_out_assign_names	Controls how entity names are handled when an object is exported to STEP format from part or assembly mode. You can give unique names to Creo Parametric datum points, datum axes, datum curves, surfaces, edges, and quilts.	no_name, user_name, id_name	no_name
Data Exchange	intf_out_auto_layer_ids	Yes - Automatically assigns interface IDs for layers that were not assigned IDs during export. No - Does not assign interface IDs for layers that were not assigned IDs during export.	yes, no	no
Data Exchange	intf_out_blanked_entities	Filters export of entities based on their blank status	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	intf_out_cat_start_model	Specifies Catia start model to be used for CATIA II .model export.		
Data Exchange	intf_out_ed_recipe_file	Sets the Creo View recipe file for export to .ed and .edz formats. For use with Windchill 8.0 and earlier servers		
Data Exchange	intf_out_layer	Layer mapping mechanism providing an ability to group entities to layers or blocks.	none, part_layer, block_layer, block_nested_layer, block_view_based_layer, block_nested_view_based_layer	none
Data Exchange	intf_out_layer_rename_table	Enables you to assign interface ids to layers during export		
Data Exchange	intf_out_max_bspl_degree	Controls the maximum degree of exported B-spline surfaces when exporting through IGES		16
Data Exchange	intf_out_pvs_recipe_file	Sets the Creo View recipe file for export to .pvs and .pvz formats. For use with Windchill 9.0 and later servers		
Data Exchange	intf_out_text_length	Sets text length in exported 2-D. As_is - Width of each character is width of strokes. Full_size - Width of each character is width of character text box. Adjusted - Adjusts spacing so no extra space left and end/start character do not overlap.	as_is, full_size, adjusted	full_size
Data Exchange	intf_pdf_u3d_create_u3d_files	YES - When exporting a Creo Parametric model as PDF U3D file, create a U3D file of the same name as the PDF file.	yes, no	no
Data Exchange	intf_profile_dir	Specifies the directory where import and export profiles are stored. Use the full path to avoid problems.		
Data Exchange	intf_pv_recipe_dir	Sets the directory for user defined Creo View recipe files		
Data Exchange	intf_ug_version	Sets the version of NX interface.	nx6, nx7, nx8	nx6
Data Exchange	intf_use_variable_size	Yes - IGES,DXF,STEP files with variable sized sheet information are imported and put on the appropriate variable-size format. Without a sheet size, the system tries to apply the correct variable-size format. No - The IGES,DXF,STEP drawing is placed on a standard sized format.	yes, no	yes
Data Exchange	intf2d_fit_incompatible_data	YES - For import and export, fixes compatibility problems between 2-D external formats (such as IGES and DXF) and Creo Parametric.	yes, no	no
Data Exchange	intf2d_iges_out_hatch	YES - Exports draft xhatching as IGES sectioned area entity. NO - exports draft xhatching as separate geometric entities.	yes, no	no
Data Exchange	intf2d_in_acad_expl_unsupported_entities	Yes - ACAD_PROXY_ENTITY and REGION entities are imported as set of draft entities, No - ACAD_PROXY_ENTITY and REGION entities are ignored.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	intf2d_in_acad_ignore_3d	YES - Ignore the 3DSOLID entities, if they exist, in the DXF/DWG file being imported in Drawing mode. Only, the 2D entities are processed. NO - Create an assembly by processing the 3DSOLID entities in the DWG/DXF file being imported in Drawing mode. This is the default value for this option.	yes, no	no
Data Exchange	intf2d_in_acad_wrap_tbl_text_s	Wrap table texts imported from a DXF or DWG file when set to YES.	yes, no	no
Data Exchange	intf2d_in_assoc_dim_geom	Controls if imported associative dimensions remain associative. Yes - If the drawing setup option associative_dimensioning is Yes, then dimensions are imported associative. No - Dimensions do not retain associativity.	yes, no	no
Data Exchange	intf2d_in_bottom_margin	Sets the bottom margin in drawing units for imported drawings.		0
Data Exchange	intf2d_in_create_multiline_note	Create a single multi-line note on importing multi-line text when is set to YES.	yes, no	yes
Data Exchange	intf2d_in_dim	AS_SEP_ENT - Imports each component of a dimension separately. AS_IS - Imports dimensions as dimensions in Pro/ENGINEER. AS_SYMBOL - Imports dimensions as symbols in Pro/ENGINEER.	as_symbol, as_is, as_sep_ent	as_is
Data Exchange	intf2d_in_dxf_mapping_file	Specifies the mapping file for DXF and DWG import. Absolute or relative path to the file can be used.		
Data Exchange	intf2d_in_iges_hatch_boundary	Yes - Places all sectioned areas boundary imported from the IGES file in one layer named IGES_HATCH_BOUNDARY. NO - Does not place sectioned areas boundary in one layer.	yes, no	no
Data Exchange	intf2d_in_iges_symbols	Controls whether to import IGES symbols as symbols or as entities by default. Yes - Import IGES symbols as symbols by default. No - Import IGES symbols as entities by default.	yes, no	yes
Data Exchange	intf2d_in_left_margin	Sets the left margin in drawing units for imported drawings.		0
Data Exchange	intf2d_in_mi_brk_unsupp_dim	Yes - Breaks unsupported MI dimension types to separate entities, No - Imports unsupported MI dimension types as different type dimensions that can look differently.	yes, no	no
Data Exchange	intf2d_in_mi_layer_parts	Yes - Maps MI drafting parts to layers, No - Imports MI drafting parts as separate entities.	yes, no	yes
Data Exchange	intf2d_in_mi_mapping_file	Specifies the mapping file for MI import. Absolute or relative path to the file can be used.		
Data Exchange	intf2d_in_mi_sym_parts	Yes - Maps MI symbol parts to symbols, No - Imports MI symbol parts as separate entities.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	intf2d_in_mi_view_parts	Yes - Maps MI annotation views to 2D views, No - Imports MI annotation views as separate entities.	yes, no	yes
Data Exchange	intf2d_in_open_log_window	Yes - Opens Import Log File in a separate window. No - The Log File does not appear in the window.	yes, no	no
Data Exchange	intf2d_in_recompute_dim_value	Indicates whether to recompute associative dimensions upon import.	yes, no	no
Data Exchange	intf2d_in_right_margin	Sets the right margin in drawing units for imported drawings.		0
Data Exchange	intf2d_in_std_clr_as_user_def	Yes - standard colors are imported as user-defined colors. No - standard colors are mapped to Creo Parametric colors.	yes, no	yes
Data Exchange	intf2d_in_top_margin	Sets the top margin in drawing units for imported drawings.		0
Data Exchange	intf2d_in_white_as_black	If set to yes entities having white color will be imported as black when the background color is white.	yes, no	yes
Data Exchange	intf2d_out_acad_brk_tbl_w_b_l_in	Yes - drawing tables having blanked lines are exported to DXF/DWG as separate entities, No - the tables are exported as DXF/DWG tables but the lines are not blanked.	yes, no	no
Data Exchange	intf2d_out_acad_mtext	Create a single MTEXT entity for multi-line text notes when is set to YES.	yes, no	yes
Data Exchange	intf2d_out_acad_ole_as_image	Controls OLE export to DXF/DWG. Yes - exports OLE objects as DXF/DWG IMAGES. No - exports OLE objects as separate entities.	yes, no	yes
Data Exchange	intf2d_out_acad_text_align	As_is - Exports notes to DXF/DWG with their original alignment values. Fit - Exports notes with FIT alignment.	as_is, fit	as_is
Data Exchange	intf2d_out_acad_unicode	Yes - Non-ASCII characters are written in UNICODE encoding.	yes, no	no
Data Exchange	intf2d_out_blanked_layers	Enables export of blanked layer entities. If an entity is placed on a layer that is blanked and this option is set to YES, the entity will be exported.	yes, no	no
Data Exchange	intf2d_out_cgm_old_lfont_map	Yes - Use the old line font mapping in CGM export: hidden lines to type 2, CTRLFONT to type 4, PHANTOMFONT to type 5. No - Use the new mapping: hidden lines to type 12, CTRLFONT to type 11, PHANTOMFONT to type 13.	yes, no	no
Data Exchange	intf2d_out_cgm_ver	Provides the ability to choose CGM metafile version when exporting from Creo Parametric drawing.	1, 3	1
Data Exchange	intf2d_out_dxf_mapping_file	Specifies the mapping file for DXF and DWG export. Absolute or relative path to the file can be used.		
Data Exchange	intf2d_out_enhanced_ents	Controls whether the enhanced entities (SPLINE and HATCHING) are converted to the corresponding DXF entities or represented as separate lines and polylines.	none, spline_and_hatch, spline_only, hatch_only	spline_and_hatch

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	intf2d_out_iges_layers_with_id	Yes - Exports interface IDs of layers as IGES levels.	yes, no	no
Data Exchange	intf2d_out_image_format	If set to "png" shaded views will be exported as a PNG image and the background will be transparent. If set to "jpeg" shaded views will be exported as a JPEG image and the background will be opaque.	jpeg, png	png
Data Exchange	intf2d_out_line_width	YES - Exports line width information to DXF. NO - all entities are exported with the default width.	yes, no	yes
Data Exchange	intf2d_out_linemerge_ignore_layers	If set to yes overlapping lines will be merged regardless of their layers when exporting or plotting a drawing. If set to no overlapping lines belonging to different layers will not be merged.	yes, no	no
Data Exchange	intf2d_out_old_quilt_priority	Yes - quilt lines have lower priority than hidden lines when exporting or plotting a drawing. No - quilt lines have higher priority than hidden lines.	yes, no	no
Data Exchange	intf2d_out_open_log_window	Yes - Opens Export Log File in special window. No - the Log File will not appear in the window.	yes, no	no
Data Exchange	intf2d_out_pnt_ent	Yes - Exports a drawing point to IGES, DXF or DWG as a point entity. No - Exports a drawing point as shape entities.	yes, no	no
Data Exchange	intf2d_out_splines_to_arcs	Yes - splines geometrically representing arcs will be exported as arcs. No - such splines will be exported as splines.	yes, no	yes
Data Exchange	intf2d_out_stroke_text	Strokes == text is converted to lines and dots. "NONE"- Does not stroke out the text for export. "ALL" - Strokes out all the text. "SPECIAL" - Strokes out only special characters.	all, none, special	none
Data Exchange	intf3d_ideas_import_filter	Enables UI to selectively import I-DEAS part and assembly models from the I-DEAS .mf1 and .pkg container files.	yes, no	no
Data Exchange	intf3d_ideas_install_dir	Specifies the path for I-DEAS installation. Use the full pathname to avoid problems.		
Data Exchange	intf3d_ideas_run_command	Specifies the I-DEAS run command. Default command is "ideas".		ideas.cmd
Data Exchange	intf3d_in_enable_layer_join	No - Join surfaces on import as geometry is loaded from file. Yes - Use new method to join surfaces by layer (surfaces contained on the same layer are joined together first) with a better chance of joining those surface elements to form closed geometry.	yes, no	yes
Data Exchange	intf3d_in_import_as_facets	Import faceted data for formats that support both facet and Brep representations.	yes, no, default	default
Data Exchange	intf3d_in_lock_redisplay	Gives the option to prevent redisplay of the model while they are in transitional state during Creo Elements Direct file import.	yes, no	no
Data Exchange	intf3d_out_anno_as	This option will determine if annotations are exported and, if exported, whether they will be exported as polyline or semantic.	none, default, graphic, semantic	graphic
Data Exchange	intf3d_out_cat2_ident_crv	Specifies model parameter IDENT_CRV while exporting to Catia.		-1

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	intf3d_out_cat2_ident_pt	Specifies model parameter IDENT_PT while exporting to Catia.		-1
Data Exchange	intf3d_out_cat2_infinity	Specifies model parameter INFINITY while exporting to Catia.		-1
Data Exchange	intf3d_out_cat2_model_sz	Specifies model parameter MODEL_SZ while exporting to Catia.		-1
Data Exchange	intf3d_out_cat2_sag	Specifies model parameter SAG while exporting to Catia.		-1
Data Exchange	intf3d_out_cat2_step	Specifies model parameter STEP while exporting to Catia.		-1
Data Exchange	intf3d_out_datums_by_default	Determines whether datum curves are included when exporting IGES files in Batch mode.	yes, no	no
Data Exchange	intf3d_out_default_option	Controls the type of 3-D data output through Pro/BATCH to an IGES or STEP file.	none, wireframe, surfaces, wireframe_surfaces, solid, shells	surfaces
Data Exchange	intf3d_out_extend_surface	Specifies how surfaces are handled for exporting files to other systems	yes, no	yes
Data Exchange	intf3d_out_force_surf_normals	IGES and PDGS export. Controls normals of non-analytic surfaces. Yes - Forces the surface normals to point in a consistent direction. No - Does not force the surface normals to point in a consistent direction.	yes, no	no
Data Exchange	intf3d_out_jt_auto_lods	Generate three default Creo Parametric LODs.	yes, no	no
Data Exchange	intf3d_out_jt_brep	No*: Export to JT as facet representation only, jt_brep: Export to JT as facet and jt_brep representation, xt_brep: Export to JT as facet and xt_brep representation	no, jt_brep, xt_brep	no
Data Exchange	intf3d_out_jt_config_name	Name of JT configuration file.		
Data Exchange	intf3d_out_jt_config_path	Path to JT configuration file.		
Data Exchange	intf3d_out_jt_structure	Specifies product structure to file structure mapping defined in JT Open Toolkit.	per_part, fully_shattered, monolithic	per_part
Data Exchange	intf3d_out_parameters	Exports parameters with models.	all, none, designated	all
Data Exchange	intf3d_out_prop_chord_heights	Use maximum chord heights proportional to component sizes.	yes, no	no
Data Exchange	intf3d_out_prop_step_sizes	Use maximum step sizes proportional to component sizes.	yes, no	no
Data Exchange	intf3d_out_surface_deviation	Sets maximum allowable deviation between original and resulting surfaces in the process of converting Creo Parametric surfaces to spline surfaces. By setting this you convert surfaces with a better approximation than by using the current model accuracy.		-1
Data Exchange	intf3d_out_unicode_to_neutral	YES - Unicode strings are exported to the Neutral file, NO - Unicode strings are converted to ascii characters for Neutral file export.	yes, no	yes
Data Exchange	intf3d_out_use_num_names	Use numeric layer names as layer export IDs.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	intf3d_out_use_step_size	Use maximum step size(s).	yes, no	no
Data Exchange	intf3d_parasolid_export_schema	Allows user to choose the schema to be used for parasolid export. Default is SCH_10004. Consult online documentation for other valid schema settings.		sch_10004
Data Exchange	medusa_2d_config_file	Specifies the Medusa 2D Interface config. file. Absolute or relative path to the file can be used.		
Data Exchange	mentor_ver_2_0	Yes - Exports the file in IDF 2.0 format. No - Exports the file in IDF 1.0 format unless IDF 2.0 is selected from the ECAD FORMAT menu.	yes, no	yes
Data Exchange	pdf_linecap	0 - Butt cap: stroked lines are squared off at the endpoint of the path. There is no projection beyond the end of the path.	butt, round, projecting square	butt
Data Exchange	pdf_linejoin	0 - Miter join: outer edges of the strokes for the two segments are extended until they meet at an angle.	round, miter, bevel	miter
Data Exchange	pdf_use_pentable	No - PDF export does not use any defined pentable changes as would be applied to plotting the drawing.	yes, no	no
Data Exchange	pro_gplug_dir	Specifies the directory in which Granite application plug-ins are located. Use the full pathname to avoid problems.		<creo3.0_loadpoint>\x86_e_win64\gplugins\
Data Exchange	put_iges_drawing_entity	Specifies whether or not to suppress the output of the IGES drawing entity, #404, to the IGES file. No - The drawing entity is not output.	yes, no	yes
Data Exchange	step_appearance_layers_groups	No - Output of Appearances, Layers, and Groups is enabled for STEP standard AP214. Yes - Output of Appearances, Layers, and Groups is enabled for STEP standard AP203.	yes, no	no
Data Exchange	step_export_ap214_asm_def_mode	Toggles the default setting of the File Structure field in the STEP Export dialog box for supported assembly STEP export files.	single_file, separate_parts_only, separate_all_objects	single_file
Data Exchange	step_export_dwg_views	AS_3D_VIEWS - Export 3-D model geometry with its associative views. AS_3D_VIEWS_ASSOC_DRAFT - Export 3-D model geometry with its associative views and view related annotations. NO - Export only 2-D representations of the 3-D models. This is the default.	no, as_3d_views, as_3d_views_assoc_draft	no
Data Exchange	step_export_format	Specifies output format when exporting to STEP. AP203_IS - Formats output to ISO 10303 AP203IS format. AP214_CD - Formats output with geometry that meets the specification for the long-term schema for AP214cc1.	ap203_is, ap202_is, ap209_dis, ap214_is, ap203_e2, ap203_is_ext	ap203_is
Data Exchange	step_out_asm_val_prop	Export validation information for assemblies to the STEP file for formats ap203_e2, ap214_is	yes, no	no
Data Exchange	step_out_material	Exports the material definition (name and density) to the STEP file for formats ap203_e2, ap214_is and 203_is_ext	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	step_out_material_as_product	Export the material definition (name and density) to the STEP file as a separate STEP PRODUCT entity	yes, no	no
Data Exchange	step_out_suppl_geom	Export datum csys, planes, axes, points to the STEP file for formats ap203_e2, ap214_is	yes, no	no
Data Exchange	system_iges_header_file	Inserts specified text file into the Start section of IGES files (refer to user_iges_header_file). When used with user_iges_header_file, the system text appears first.		
Data Exchange	tiff_compression	TIFF exports are done without compression.	none, g4, packbits, deflate	none
Data Exchange	tiff_type	Determines the type of tiff items are exported to variables refer to color sets	rgb, palette, grayscale, mono	rgb
Data Exchange	use_export_2d_dialog	Yes - Opens export options dialog when exporting Creo Parametric drawing. No - exports file without the options dialog.	yes, no	yes
Data Exchange	use_iges_font_1003	Used to disable the use of IGES font 1003.	yes, no	yes
Data Exchange	use_iges_kanji_font_2001	Specifies whether to convert Kanji notes in Creo Parametric to IGES Kanji notes (font code) when exporting. Yes - Converts Kanji notes to IGES Kanji notes. No - Converts using Font 1.	yes, no	no
Data Exchange	user_iges_header_file	Filename - Insert specified text file into the Start section of IGES file. Valid parametric note symbols will be replaced during export. For example, &dwg_name in the text file is replaced by the drawing name when a drawing is exported.		
Data Exchange	vda_header	The complete filename of the text file containing VDA header information. If you intend to use the same header for all VDA files, specify the full pathname.		
Data Exchange	vrml_anchor_url	Enables you to place an anchor on a specified VRML component during the export to VRML. The keywords are optional.		
Data Exchange	vrml_background_color	Yes - Exports a model to VRML with the Creo Parametric background color.	yes, no	no
Data Exchange	vrml_explode_lines	Yes - Exports a model to VRML with explode lines for assembly or assembly process data.	yes, no	yes
Data Exchange	vrml_export_resolution	Specifies the number of levels of detail (LODs) in models exported in VRML format.	high, medium, low	medium
Data Exchange	vrml_export_version	Allows the user to choose between exporting to the VRML 2.0 or 1.0 format.	2.0, 1.0	2
Data Exchange	vrml_file_duplicate_material	Yes - Ensures that the model components retain their true color. No - In some viewers, component colors might not be consistent.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Exchange	vrmf_multiple_views	All - Exports top-level and lower-level assembly component views into VRML format. None - Does not export component views into VRML format. Top - Exports top-level object view only into VRML format.	all, none, top	all
Data Exchange	vrmf_parameters	Controls the export of user parameters. Designated - Export only designated parameters. All - Export all parameters. None - Do not export parameters.	all, none, designated	designated
Data Exchange	vrmf_simpref_export	Yes - Specifies direct and in memory export of top-level assembly simplified representation to Pro/FLY-THROUGH package files.	yes, no	no
Data Exchange	www_add_aux_frame	Creates additional frame for each process step or for assembly publication. Yes - Auxiliary files aux.html are created for each step in assembly process (in each step00 directory) to be replaced by your .html file. No - Auxiliary files are not created.	yes, no	no
Data Exchange	www_export_geometry_as	Specifies the output format by to one of the values	all, vrmf, cgm, jpg, cgm_vrmf, jpg_vrmf, cgm_jpg	jpg_vrmf
Data Exchange	www_multiple_views	All - Writes all views in any component to the appropriate VRML files. Top - Writes only named views that reside in the assembly or process assembly to top-level VRML files. None - Does not write named views to VRML files.	all, none, top	top
Data Exchange	www_tree_location	Specifies the location of the model tree in the browser window. Out - Opens model tree in a separate window. In - Includes model tree in the Web page and removes the Tree checkbox from the control panel.	out, in	out
Data Management	allow_import_file_extension	Allow additional files with extensions to be shown in the import dialog		
Data Management	bitmap_size	Works in conjunction with save_bitmap. Determines the size (squared) of the bitmap image that is stored.		200
Data Management	dm_auto_open_zip	YES - Windchill CAD packages or zip files will open automatically. NO - User selects which file in contents to open.	yes, no	yes
Data Management	dm_cache_limit	Enter amount of disk space (in megabytes) to allocate for local file storage. The change to the existing setting will take effect after Creo Parametric is restarted		
Data Management	dm_cache_mode	Indicate which objects will be written to local cache when objects in Creo Parametric memory are saved.	all, none, modified	all
Data Management	dm_checkout_on_the_fly	Controls the default action presented to the user when Check Out on the fly dialog comes up	checkout, continue	checkout

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Management	dm_fallback_server_location	Specifies the server location where objects will be saved when Creo Parametric does not provide an ability to specify the target save location. This option is applicable for Windchill ProductPoint only.		
Data Management	dm_hide_virtual_default_ws	Yes - When there is at least one workspace created in any context in selected server, default workspaces that were never used, will not be listed. No - (Default value) Default workspaces will be listed for each context that has no workspace defined.	yes, no	no
Data Management	dm_http_compression_level	Set the data compression factor (0-9, 0= no compression) for data exchange with a Windchill server.		
Data Management	dm_offline_after_event	Allow the user to choose to work offline when there is a loss of server connection. Staying online will continue to retry server operations.	yes, no	yes
Data Management	dm_offline_options_activated	Yes - Synchronize, Download and Upload boxes are checked when going online. No - Only Synchronize box is checked when going online.	yes, no	yes
Data Management	dm_overwrite_contents_on_update	Yes: Windchill Workspace Update will overwrite locally/modified out of date objects with the ones in the server. No (Default): Windchill Workspace Update will update metadata only.	yes, no	no
Data Management	dm_remember_server	Primary server/workspace must be set by user for each Creo Parametric session.	yes, no	yes
Data Management	dm_save_as_attachment	Controls the default option for Save A Copy when models are saved as in non-Creo Parametric format. Yes - It will by default save it as secondary content attachment to the original CAD Document. No- It will by default save it as Primary CAD Document	yes, no	yes
Data Management	dm_search_primary_server	Search the primary server for dependencies not found in the workspace.	yes, no	yes
Data Management	dm_secondary_upload	Indicate when modified Creo Parametric objects on a secondary server will be moved from the local cache to the user's workspace on the server.	automatic, explicit	automatic
Data Management	dm_synchronize_in_background	Yes: Enables workspace synchronization in the background. No (Default): Workspace synchronization will happen in the foreground.	yes, no	no
Data Management	dm_update_unconfirmed_instances	Specifies when to force verification and save of unconfirmed assembly instances to data management system: NEVER - never; CRITICAL_ONLY - only to correct or prevent doubtless data corruption; ALWAYS - whenever generic is saved	never, always, critical_only	never
Data Management	dm_upload_objects	Indicate when modified Creo Parametric objects will be moved from the local cache to the user workspace on the server.	automatic, explicit	explicit

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Data Management	let_proe_rename_pdm_objects	Determines whether an object fetched from a PDM database can be renamed in a Creo Parametric session.	yes, no	no
Data Management	pdm_rev	Specify how the system parameter, PDM_REV, should be formatted. Only useful for objects submitted to Pro/PDM or Pro/INTRALINK.	pdmrev_fmt_rev_ver, pdmrev_fmt_rev, pdmrev_fmt_branch_rev_ver, pdmrev_fmt_branch_rev	pdmrev_fmt_rev_ver
Data Management	read_parameters_of_excluded	Allow Creo Parametric designated parameters on excluded objects in Simplified Representation to be visible in Model Tree columns.	yes, no, common_name_only	no
Data Management	regenerate_read_only_objects	Determines whether an object retrieved from PDM database with read only status can be regenerated in a Creo Parametric session.	yes, no	yes
Data Management	save_bitmap	When performing a save on a model, determines what level of the assembly will have bitmap images created of modified components.	none, toplevelonly, alllevels	none
Data Management	save_bitmap_type	When performing a save on a model, determines the type of image file that is created for that model. Used with "save_bitmap" option.	cgm, bmp, tiff, gif, jpeg	bmp
Data Management	save_unchanged_pdm_object	Determines how to save in the current working directory an object fetched from a Pro/PDM database into a Creo Parametric session.	as_ref, as_copy	as_ref
Design Manager	dont_show_symbol_nodes	Do not show references derived from relations in Parent/Child viewer.	yes, no	no
Dimensions & Tolerances	allow_rfs_default_gtols_always	Yes - Enables RFS/Default gtols to be created even when not permitted by ANSI standard.	yes, no	no
Dimensions & Tolerances	ang_dim_in_screen	Yes - If the display is in the default pan/zoom setting (View>Pan/Zoom>Reset), the system checks whether an angular dimension is visible on the screen. If it is not visible, then the system moves the dimension to a visible location.	yes, no	no
Dimensions & Tolerances	angular_tol	An alternative format for setting default angular tolerance dimensions. The value sets the number of decimal places, and the tolerance is the actual tolerance value. For integer dimensions, the value is zero, and the tolerance is an integer.		0
Dimensions & Tolerances	angular_tol_0.0	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.		5
Dimensions & Tolerances	angular_tol_0.00	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.		50
Dimensions & Tolerances	angular_tol_0.000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.		500

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Dimensions & Tolerances	angular_tol_0.0000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.		5000
Dimensions & Tolerances	angular_tol_0.00000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.		50000
Dimensions & Tolerances	angular_tol_0.000000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.		500000
Dimensions & Tolerances	create_fraction_dim	Yes - All dimensions created will be displayed as fractions.	yes, no	no
Dimensions & Tolerances	default_ang_units	Sets the display of newly created angular dimensions both in the model and in the drawing. There is no impact on dimensions previously created. ang_deg - Decimal degrees. ang_min - Degrees and decimal minutes ang_sec - Degrees, minutes, and decimal seconds	ang_deg, ang_min, ang_sec	ang_deg
Dimensions & Tolerances	default_dec_places	Sets the default number of decimal places (0-13) to which to round newly created dimensions. Unrounded dimensions automatically determine the number of decimal places required to display their stored value.		2
Dimensions & Tolerances	default_dim_num_digits_changes	Sets the default number of digits displayed in a dimension to the last entered value. No - The system defaults to the value specified for the configuration file option default_dec_places.	yes, no	yes
Dimensions & Tolerances	default_placement_surfacefinish	Sets the default placement type for surface finishes within the surface finish UI.	with_leaders, on_entity, normal_to_entity, free	normal_to_entity
Dimensions & Tolerances	dim_fraction_denominator	Sets the largest denominator to be used for fractional dimensions. If the fraction can be reduced, then it converts to the lowest possible denominator (for example, 4/32 converts to 1/8).		32
Dimensions & Tolerances	dim_fraction_format	Std - Displays fractional dimensions in the standard Creo Parametric format. Aisc - Displays fractional dimensions in the American Institute of Steel Construction (AISC) format and displays architectural units in AISC format for feet-inches.	std, aisc	std
Dimensions & Tolerances	dim_offscreen_limit	If more than zero, sets the margin beyond which dimensions are automatically snapped back onto the screen. For example, a value of .2 lets a dimension be 20% of the window size outside the window. If set to zero, disables snapping of offscreen dimensions.		0.2
Dimensions & Tolerances	display_tol_by_1000	If set to Yes, tolerances for non-angular dimensions will be displayed multiplied by 1000.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Dimensions & Tolerances	gtol_dim_placement	Controls display of geometric tolerances in Part. Use like drawing set-up option of the same name. On_bottom - Gtols display at bottom (below additional lines of text). Under_value - Gtols display under dimension value (above additional lines of text).	on_bottom, under_value	on_bottom
Dimensions & Tolerances	linear_tol	An alternate format for setting default linear tolerance dimensions. First value sets the number of decimal places. Second value is the actual tolerance. For example, 6 0.000025 sets tolerance to six decimal places. The default tolerance is 0.000025.		0.000000 -1
Dimensions & Tolerances	linear_tol_0.0	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.		1
Dimensions & Tolerances	linear_tol_0.00	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.		1
Dimensions & Tolerances	linear_tol_0.000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.		1
Dimensions & Tolerances	linear_tol_0.0000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.		1
Dimensions & Tolerances	linear_tol_0.00000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.		1
Dimensions & Tolerances	linear_tol_0.000000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.		1
Dimensions & Tolerances	maintain_limit_tol_nominal	Maintains the nominal value of a dimension regardless of the changes that you make to the tolerance values.	yes, no	no
Dimensions & Tolerances	mark_approximate_dims	Yes - Displays a tilde (~) in front of a dimension that is set to display as a fraction and has an actual value that is not rounded to equate exactly to the displayed fractional value (the fraction shows 7/32 but the actual value is .22, not 0.21875).	yes, no	no
Dimensions & Tolerances	minimum_angle_dimension	Controls the minimum angle value under which Creo Parametric assumes colinearity of the selected items when creating a driven dimension.		1
Dimensions & Tolerances	parenthesize_ref_dim	Encloses reference dimensions in parentheses. No - Follows reference dimensions with the text "REF".	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Dimensions & Tolerances	restricted_gtol_dialog	Determines whether the Geometric Tolerance dialog restricts the user by graying out elements that are considered "illegal".	yes, no	yes
Dimensions & Tolerances	round_displayed_dim_values	Sets the default status of the Rounded Dimension Value checkbox for newly created or shown dimensions.	yes, no	yes
Dimensions & Tolerances	tbl_driven_tol_val_edit	Tbl_driven_tol_val_edit Y/N* - "Yes" allows users to directly edit the tolerance value of dimensions whose tolerance values are driven by tolerance tables. Editing the tolerance value of a dimension will make the dimension non-table-driven. "No" disallows direct editing of the tolerance values of table driven tolerances.	yes, no	no
Dimensions & Tolerances	tol_display	Displays dimensions with or without tolerances.	yes, no	no
Dimensions & Tolerances	tolerance_class	Sets the default tolerance class for ISO-standard models. The system uses the tolerance class in conjunction with the dimension value when retrieving tolerances for General or Broken Edge dimensions.	medium, fine, coarse, very_coarse	
Dimensions & Tolerances	tolerance_standard	Sets the tolerance standard used when creating the model.	ansi, iso	ansi
Dimensions & Tolerances	tolerance_table_dir	Sets the default directory for user-defined tolerance tables for ISO-standard models. All Holes and Shafts tables overwrite existing tables when loaded.		
Dimensions & Tolerances	use_major_units	Determines if fractional dimensions are displayed in terms of feet-inches or meter-mm. Yes - The major units are used. For example, when the units are inches and 25.125 is converted to a fraction, the dimension becomes 2' 1-1/8".	yes, no	no
Dimensions & Tolerances	use_nom_dim_val_in_expr	Yes - use dimensions' nominal values in expressions. No - use current values.	yes, no	no
Drawing	allow_move_attach_in_dtl_move	Yes - Move and Move Attach commands in drawing mode act together. No - Move and Move Attach commands in drawing mode do not act together.	yes, no	yes
Drawing	allow_move_view_with_move	Disallow the movement of drawing views with the mouse.	yes, no	no
Drawing	allow_refs_to_geom_reps_in_drws	Yes - Allows you to create drawing references to geometry representations (includes dimensions, notes, and leaders). These references may become invalid if the referenced geometry changes resulting in geometry representations not updating in drawings.	yes, no	no
Drawing	allow_vert_select_in_dim_create	Allow highlight of model vertices during dimension placement in drawings	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Drawing	auto_constr_offset_tolerance	Sets the auto constraint tolerance for creating offset dimensions. If distance is less than this tolerance multiplied by component size, offset is set as coincident. Default value is 0.5 .		0.5
Drawing	auto_regen_views	Yes - Automatically repaints the drawing display when changing from one window to another. No - Update to drawing views will only occur when manual updates are made via Repaint or View> Update.	yes, no	yes
Drawing	autobuildz_enabled	Loads AutobuildZ application.	yes, no	no
Drawing	bom_format	Sets the BOM format file to be used for a customized BOM. Specify name and path.		<creo3.0_loadpoint>\text\format.bft
Drawing	create_drawing_dims_only	Yes - Saves all new driven dimensions created in the drawing inside the drawing as associative draft dimensions. No - Saves driven dimensions created in Drawing mode on solid geometry in the solid.	yes, no	no
Drawing	default_ang_dec_places	Sets the default number of decimal places (0-13) to which to round newly created angular dimensions. Unrounded angular dimensions automatically determine the number of decimal places required to display their stored value.		1
Drawing	default_draw_scale	Sets the default drawing scale for views added with the No Scale command. The value must be greater than 0. No - The system does not set a default drawing scale.		-1
Drawing	default_font_kerning_in_drawing	Determines initial setting of Font kerning when 2D drawing annotations are created. Yes - Font kerning is on for new 2D annotations.	yes, no	no
Drawing	disp_trimetric_dwg_mode_view	Displays the model in default orientation when placing a general view on the drawing. No - Model does not appear until Default is chosen from the Orientation dialog box.	yes, no	yes
Drawing	display_dwg_sketch_constraint	When set to yes, parametric sketching constraints, such as V for vertical, are displayed when a drawing object is selected	yes, no	no
Drawing	display_dwg_tol_tags	Sets the display of the tolerance tags that appear in drawings. This option does not affect the display of tolerances on dimensions.	yes, no	yes
Drawing	display_in_adding_view	Controls model display in a drawing when auto_regen_views is No. Wireframe - Adds new views in wireframe, displays datums. Minimal_wireframe - Displays in wireframe, no datums, axes, or silhouette edges. Default - Displays using the Environment setting.	default, wireframe, minimal_wireframe	wireframe
Drawing	draw_models_read_only	Yes - Drawing model files are read-only. No model changes can be made. Cosmetic_only - Only cosmetic changes can be made to the model. Design changes are not allowed. Cosmetic changes include adding driven dimensions, cross sections, and explode states.	yes, no, cosmetic_only	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Drawing	draw_points_in_model_units	Defines the current draft view's coordinate values as model units rather than drawing units. The GET POINT menu uses the scale of the draft view and the draft view's model units for relative and absolute coordinate entry and display in the Message Window.	yes, no	no
Drawing	drawing_ole_image_dpi	Resolution of OLE object images. The range of valid values is 0 to 600 dots per inch.		150
Drawing	drawing_setup_file	Sets the default drawing setup file option values for your Creo Parametric session, otherwise, the system uses the default drawing setup file option values. Certain parameters in the file are valid only if you have a license for Pro/DETAIL.		<creo3.0_loadpoint>\text\prodetail.dtl
Drawing	drawing_shaded_view_dpi	Resolution of exported shaded drawing views. The range of valid values is 0 to 600 dots per inch.		150
Drawing	drawing_view_origin_csys	The named coordinate system will be used as the origin of a newly created view or of a modified view. None - The system will not use a previously specified coordinate system.	none	
Drawing	drawing_warn_if_flex_feature	Display a warning dialog upon opening a drawing with models that contain flexible features advising users that annotations may be out of date and should be verified.	yes, no	yes
Drawing	dwg_unicode_conversion_language	Defines the language used during Unicode conversion of pre-Wildfire 4.0 drawings. This option does not affect the conversion of models. File - Uses the language stored in the drawing header file; Session - Uses the language of the current Creo Parametric session.	file, session, english, japanese, french, german, russian, italian, spanish, korean, chinese_tw, chinese_cn, hebrew, greek, turkish, czech, polish, hungarian, slovenian, portuguese, slovak, usascii	file
Drawing	enable_shaded_view_in_drawings	Yes - shading display is allowed in drawing views. No - shading display will be shown in wireframe.	yes, no	yes
Drawing	force_wireframe_in_drawings	Yes - Displays all views of drawings in wireframe. No - Displays views according to the display set for the session (wireframe, hidden, and so on).	yes, no	no
Drawing	format_setup_file	Assigns a specified setup file to each drawing format. To assign the drawing's parameter values to the format, you must retrieve the drawing's setup file into the format.		dwgform.dtl
Drawing	highlight_erased_dwg_views	Controls the display of erased view outlines (environment option Highlight Erased Views). Yes - Highlights erased views. No - Turns off highlighting for erased views.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Drawing	highlight_new_dims	In drawing mode, highlights new dimensions in red until you repaint the screen.	yes, no	no
Drawing	hlr_for_quilts	In Drawing, controls quilt display in hidden line removal. Yes - Includes quilts in the hidden line removal process (preselects Qlt HLR in the VIEW DISP menu). Default in the VIEW DISP menu, uses the value you have set.	yes, no	no
Drawing	make_aux_view_notes	Automatically adds view names to auxiliary views using AUX VIEW* viewname-viewname. You can modify the view name after adding the view to the drawing.	yes, no	no
Drawing	make_parameters_from_fmt_tables	Determines the handling of values entered in a format table when you replace one drawing format with another. Yes - Stores entered values for format table and copies them with the table. No - Prompts you to re-enter all the values for the format table.	yes, no	no
Drawing	make_proj_view_notes	Automatically adds view names to projection views in the format "VIEW viewname-viewname." You can modify the view name after adding the view to the drawing.	yes, no	no
Drawing	ole_unlimit_zoom	Yes - The OLE object is displayed in any zoom level (could result in performance degradation). No - The OLE object disappears in high zoom-in levels.	yes, no	no
Drawing	pick_chain_tangent_only	Specifies the extent of draft entity chains selected. Yes - Only entities in the chain that are tangent will be selected. No - All entities connected end to end will be selected.	yes, no	no
Drawing	preferred_export_format	Set the default preferred export file type for export from drawings.	cgm, tiff, iges, dxf, step, pdf, dwg, medusa, stheno	pdf
Drawing	pro_dtl_setup_dir	Sets the directory for the drawing setup files. If this option is not set, the system uses the default setup directory.		
Drawing	pro_format_dir	Sets the default directory for the drawing format library. Use the full pathname to avoid problems.		
Drawing	pro_note_dir	Specifies the directory from which to retrieve notes entered from a file. Use the full path name to avoid problems.		<dir_name>\dump_x86e_win64\
Drawing	pro_palette_dir	Sets the default directory for drawing symbol palette files.		<creo3.0_loadpoint>\symbols\palette\
Drawing	pro_symbol_dir	Set and automatically create the default directory for saving and retrieving user-defined symbols. If you do not specify a directory, the system uses the current working directory. Use full path name to avoid problems.		<dir_name>\dump_x86e_win64\
Drawing	pro_table_dir	Specifies the directory to be used when tables are retrieved or saved. Use the full path name to avoid problems.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Drawing	remember_last_get_point_option	Yes - Sets the default of the GET POINT menu as the last option picked among Pick Pnt, Vertex and On Entity.	yes, no	no
Drawing	save_display	Yes - Stores view geometry and detail items such as solid dimensions. These items are displayed when retrieving the drawing in View-Only mode.	yes, no	no
Drawing	save_drawing_picture_file	Embed - Embeds a picture file inside a drawing for preview purposes. Export - Saves a drawing file as a picture file in the working directory when saving a drawing. Both - Does both embed and export.	no, export, embed, both	embed
Drawing	save_modified_draw_models_only	Determines whether the system saves the model after you have changed it. No - Saves the model every time that you store the drawing.	yes, no	yes
Drawing	select_hidden_edges_in_dwg	Controls selectability of hidden edges in no-hidden drawing views.	yes, no	yes
Drawing	selection_of_removed_entities	Yes - Entities in front of cross-section (planar or offset) can be selected, clipped (using Z-clipping), or erased with EDGE DISP menu. No - Geometry, datum points or curves, cosmetic features, threads, grooves, or coordinate systems cannot be erased.	yes, no	no
Drawing	show_annotations_dialog_limit	Default value for Annotations dialog box.		
Drawing	show_preview_default	This option determines the default behavior for preview in Show/Erase.	remove, keep	remove
Drawing	snap_drw_entity_to_snap_line	Yes - Enable snapping of drawing and report entities to snap lines. No - disable snapping of drawing and report entities to snap lines.	yes, no	yes
Drawing	switch_dims_for_notes	Yes - Dimensions will be displayed in their symbolic format during drawing note creation. No - Dimensions will be displayed in drawing notes as numerical values.	yes, no	yes
Drawing	sym_leader_orient_move_text	With Pro/DETAIL, automatically regroups weld symbol instances after you move the text.	yes, no	no
Drawing	symbol_instance_palette_file	Specifies the location of the symbol instance palette.		
Drawing	symbol_palette_input	Controls the display of the special symbol palette during note creation.	yes, no	yes
Drawing	todays_date_note_format	Controls the initial format of the date displayed in a drawing. The format for the setting is a string consisting of three portions: the year, the month, and the date. You can enter the portions in any order.		%dd-%mmm-%yy

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Drawing	tol_num_digits_default_drive n	Yes - The Default check boxes in the number of digits area of the Dimension Properties dialog will be checked when a dimension is created or first shown. No - The check boxes will be unchecked when a dimension is created or first shown.	yes, no	yes
Drawing	triangulate_filled_areas	Subdivide filled areas into triangles (may affect memory usage and plot files).	yes, no	no
Drawing	variant_drawing_item_sizes	No - Drawing items moved/copied to a different sheet or located on a changed sheet keep same size and relative orientation on paper. Yes - Some items scale/reposition to be the same on paper, and others scale and/or reposition to be the same on screen.	yes, no	no
Drawing	warn_if_iso_tol_missing	Yes - Returns the user to the dimension properties dialog to select a different table. No - Applies the tolerance according to existing functionality.	yes, no	no
Education & Social Tools	enable_learning_connector	Enable the Learning Connector, which provides context specific access to videos, training, help topics, and technical support content. When enabled, the Learning Connector is added to the ribbon bar next to Help.	yes, no	yes
Electromechanical	align_cable_bundles	Align NEWLY CREATED bundles where they meet or branch out	yes, no	yes
Electromechanical	allow_bundle_flat_cables	Allow to create a bundle over a group of flat packing cables.	yes, no	no
Electromechanical	allow_harn_mfg_assy_retrieval	Allow to retrieve a harness mfg assembly directly.	yes, no	no
Electromechanical	assemble_parent_connector	Assemble parent connectors on locations referencing a subconnector.	yes, no	yes
Electromechanical	auto_convert_cables	Yes - When regenerating a pre-Release 11.0 cabling assembly, the system automatically creates a separate spool feature for every individual wire or cable in the assembly. No - An interface for the cable conversion appears.	yes, no	yes
Electromechanical	auto_xml_on_retrieve	Automatically load Cabling Logical Reference from an XML file when retrieving the cabling assembly	yes, no	no
Electromechanical	auto_xml_on_save	Automatically create an XML Logical Reference when saving a cabling assembly	yes, no	no
Electromechanical	autoroute_path_param_name	Set the wire parameter to use to select a path during autorouting.		use_path
Electromechanical	boeing_extract_bundle	Yes - Uses the extract bundle algorithm. No - Does not use the extract bundle algorithm.	yes, no	no
Electromechanical	bundle_transition_create_method	Control whether bundle transitions are created automatically. Default is automatic.	automatic, manual	

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Electromechanical	cable_bundle_transition_length	Transition length is controlled by the formula Larger Diameter X <percentile>. Default value is 50±0...250]%.</td>		
Electromechanical	cable_int_portions_for_clr	No - Global clearance checking for internal cable portions is excluded. Yes - Global clearance checking for internal cable portions is included.	yes, no	no
Electromechanical	cable_location_override_color	Override the system color of cable locations. Enter RGB values between 0 and 255 separated by space.		
Electromechanical	cable_segment_override_color	Override the system color of cable segments that do not have a color map. Enter RGB values between 0 and 255 separated by space.		
Electromechanical	cables_with_hlr	Cables are not hidden behind objects.	yes, no	yes
Electromechanical	cabling_split_multi_pnt_locs	Setup whether to convert multiple-location features into single location features.	yes, no	yes
Electromechanical	compact_cables_packing	Setup whether the new packing of cables should be applied.	yes, no	no
Electromechanical	compare_spools_by_name	Compare spools by names in logical compare.	yes, no	yes
Electromechanical	cti_class_color_file	Sets the CTI class color file for CTI class color display during electrical clearance and creepage analysis. Use the full pathname to avoid problems.		
Electromechanical	diagram_export_allow	YES: Allow user to save a copy of loaded diagrams as DGE file for loading into Routed Systems Designer, NO: Do not provide option to save a diagram as DGE.	yes, no	no
Electromechanical	diagram_pin_edit_multi_line	(Diagram) Determines the Pro/TABLE format used when you are modifying pin parameters. "yes" - You can add user-defined pin parameters by including them between the DEFINE and ENDDF statements for each pin. "no" - The system uses columnar format and	yes, no	no
Electromechanical	display_duplicate_harness_msg	Display warning message when duplicate harness are identified in the same assembly.	yes, no	yes
Electromechanical	display_internal_cable_portion	Yes - Internal cable portions will be displayed for any wires that have been manually set to Yes in the Cable Paths Setup dialog box for each individual splice and custom component. This option can be overridden in the Environment dialog box.	yes, no	no
Electromechanical	display_route_cable_preview	If no then Route Cables tool does not display preview of cables as they are routed. Otherwise Route Cables tool will show a preview of the path routed cables will follow.	yes, no	yes
Electromechanical	display_thick_cables	Yes - Display cables and wires with thickness. No - Display the centerline of cables and wires only. This option can be overridden in the Environment dialog box.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Electromechanical	enable_cable_channel	Activates channel functionality	yes, no	no
Electromechanical	fan_significant_locations_only	Setup whether only significant locations should be flattened on flattening a harness.	yes, no	yes
Electromechanical	fan_with_network	Setup whether the network should be included on flattening a harness.	yes, no	yes
Electromechanical	freeze_failed_cable_locations	Suspends the cable location at its last known assembly location.	yes, no	no
Electromechanical	full_hlr_for_cables	Full - Removes hidden lines from view when cables hide other geometry (use when Hidden Line active). Partial - Cables hide other non-interfering cables except when cables route together between same locations. None - Display hidden lines (faster).	none, full, partial	full
Electromechanical	harn_start_model_dir	Provides the complete path to the directory containing harness start parts and assemblies		
Electromechanical	harn_tang_line_display	Display the tangency lines between cable segments in thick cable display mode.	yes, no	yes
Electromechanical	hlr_for_xhatches	Controls whether HLR to be enabled for crosshatches in drawing mode.	yes, no	no
Electromechanical	ignore_bundles_in_find_cables	Include bundles and overbraids in the query results when the model or logical data is queried for unrouted items.	yes, no	no
Electromechanical	network_location_override_color	Override the system color of network locations. Enter RGB values between 0 and 255 separated by space.		
Electromechanical	network_segment_override_color	Override the system color of network segments. Enter RGB values between 0 and 255 separated by space.		
Electromechanical	orthogonal_snap	Controls non-90 degree routing in Pro/DIAGRAM. Yes - Turns the Ortho Snap function on. You can sketch wires only at the default horizontal and vertical orientation. No - Enables you to sketch wires in drawings at angles other than the default.	yes, no	yes
Electromechanical	pro_cbltrm_dir	Sets the directory from which terminators are retrieved. Use the full pathname to avoid problems.		
Electromechanical	pro_spool_dir	Sets the directory from which spools are retrieved. Use the full pathname to avoid problems.		<dir_name>\dump_x86e_win64\
Electromechanical	report_end_splice	Setup whether the terminal splices of cables are reported.	yes, no	no
Electromechanical	ribbon_cable_flat_direction	Flat ribbon cables in the y-axis of entry ports.	y_axis, x_axis	y_axis
Electromechanical	smaller_bundle_min_bend_radius	Setup whether to consider the thickness of the bundle when set up the minimum bend radius.	yes, no	no
Electromechanical	template_flat_harness	Specify the model used as the default flattened Harness assembly template.		
Electromechanical	template_harnesspart	Specify the model used as the default Harness part template.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Electromechanical	terminator_table_out_unicode	YES - Unicode strings are exported in terminator table files, NO - Unicode strings are converted to ASCII characters for terminator table export.	yes, no	yes
Electromechanical	trace_thru_splice_connector	Report the from/to connectors of cables passing through splices correctly.	yes, no	no
Electromechanical	update_harness_mfg_line_color	Update cable line color for harness-mfg part created in release 2000i or earlier.	yes, no	no
Electromechanical	update_loom_bundle_data	Setup whether to update loom bundle data from old harnesses.	yes, no	no
Electromechanical	update_pre_130_harness	Yes - When a cable is added between two previously laid out locations in a three-dimensional reference harness, display of the new cable will update automatically in pre-Release 13 flattened cable models.	yes, no	no
Electromechanical	update_pre_16_cable_layers	Yes - Harnesses containing cables routed prior to Release 16.0 will be automatically updated upon retrieval to comply with new display standards.	yes, no	no
Environment	accuracy_lower_bound	Enter an accuracy value to override the default lower limit. The minimum value for relative accuracy is 1E-06.		0.0001
Environment	activate_window_automatically	Yes - Execute Activate command automatically after a window is selected. No - Don't execute Activate command automatically after a window is selected.	yes, no	yes
Environment	bell	Yes - Turns on the keyboard bell that rings after each prompt. No - Turns off the keyboard bell. To override this setting use the Environment dialog box on the Tools menu.	yes, no	no
Environment	clearance_triangulation	Specifies the quality of surface triangulation used to obtain initial guess points for clearance and interference calculations.	none, high, medium, low	none
Environment	clock	Yes - Displays the clock while Creo Parametric is working.	yes, no	no
Environment	clr_print_plus_minus	Sets the display of the system accuracy when calculating clearances between two parts or surfaces. The default is to not display the accuracy.	yes, no	no
Environment	company_name	Used with the export option iges_out_mil_d_28000. To support MIL-D-28000, your company name is required input and the system prompts you every time you export through IGES unless you set the company name in your configuration file.		
Environment	default_abs_accuracy	Defines the default absolute part accuracy.		0
Environment	des_exp_confirm_on_save	When you use a `Save` command during a Design Exploration session, it does not save the session, it prompts to add a new checkpoint.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Environment	des_exp_create_auto_cps	Set the maximum number of automatically created checkpoints allowed. Additional checkpoints override previous checkpoints. Type a value from 1 to 99. Type 0 to prevent automatic checkpoint creation.		
Environment	des_exp_create_backups	Back up all models in session when starting Design Exploration and all models opened and modified during the session. When backup is on, starting a Design Exploration session might take some time.	yes, no	no
Environment	des_exp_data_path	Sets the directory for the Design Exploration session. The default location is in a default system folder. Type a valid file path to change the directory to a user defined one. Data from only one session is stored.		
Environment	des_exp_display_type	Set the default display of the Checkpoint Tree. 'Tree' - Displays all branches in the Checkpoint Tree. 'Timeline' - Displays checkpoints in chronological order (des_exp_display_type).	tree, timeline	tree
Environment	des_exp_show_cp_props_dialog	Open the Checkpoint Properties dialog box when you add a new checkpoint, or turn an automatic checkpoint into a regular one.	yes, no	yes
Environment	display_annotations	Determines whether or not annotations are displayed in 3D models.	yes, no	yes
Environment	display_full_object_path	Controls the display of the object's full file name (including its object-type suffix and its version number) and file path in the window title and Model Info display. Yes - Displays full file name and file path. No - Displays object name only.	yes, no	no
Environment	display_model_anls_selection	Determines whether or not to highlight the selected components during a Model Analysis.	yes, no	no
Environment	enable_absolute_accuracy	Controls display of the ACCURACY menu. Yes - ACCURACY menu always appears when you choose Accuracy from the PART SETUP menu. No - The menu appears only if the part is currently defined with absolute accuracy.	yes, no	no
Environment	flip_arrow_scale	Sets a scale factor for enlarging the size of the flip arrow that appears for feature creation direction (default = 1).		1
Environment	highlight_geometry	Yes - Items selected in the model tree are highlighted in the geometry area.	yes, no	yes
Environment	info_output_format	Setup the default format type for information presentation. Text - information will be output as simple text. HTML - information will be output as html data.	text, html	html
Environment	info_output_mode	Sets default method for presenting information. Both - View information on screen and write information to file. Screen - View information on screen only. File - Write information to file only. Choose - Select method from the INFO OUTPUT menu.	both, choose, screen, file	both

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Environment	intelligent_fastener_enabled	Enables Intelligent Fastener. You must restart Creo for the setting to take effect.	yes, no	yes
Environment	mass_property_calculate	Calculate mass properties upon regeneration, upon save or by user request	automatic, by_request, check_upon_save	by_request
Environment	mathcad_automapping	Toggles Mathcad analysis automapping of variables	yes, no	no
Environment	mdx_collision_num_triangles	The number of mesh triangles which are used to approximate model in collision detection.		1000
Environment	measure_sig_figures	Sets the number of significant figures displayed as results when using the Measure dialog box from the Info menu.		6
Environment	model_grid_balloon_display	Controls the display of grid balloons.	yes, no	yes
Environment	model_grid_neg_prefix	Enables you to specify the prefix to use for negative numbers in the balloon capture of the grid.		-
Environment	model_grid_num_dig	Controls the number of decimal places displayed in the balloon capture. 0 - Only integers display.		
Environment	model_grid_spacing	Sets the default model grid spacing.		1.5
Environment	model_notes_as_labels	Gives the option to display notes as either full text or as label. Can also be toggled with environment option "Display Notes as Names."	yes, no	no
Environment	model_tree_start	Toggles display of the Model Tree. Yes - Displays the Model Tree window along with its model. No - Does not display the Model Tree window.	yes, no	yes
Environment	modelcheck_enabled	Enable ModelCHECK	yes, no	yes
Environment	mp_auto_update_precreo_models	no - Creo does not recalculate the mass properties of legacy parts upon retrieval of a model. yes - Creo recalculates mass properties of legacy parts upon retrieval of a model except for legacy parts with no assigned density.	yes, no	no
Environment	part_mp_calc_ignore_alt_mp	Yes (default) - Creo uses the calculated mass (mass=volume*density) to calculate part mass properties. No - Creo uses the user-defined value of PRO_MP_ALT_MASS if the value of PRO_MP_SOURCE is Geometry and Parameters or Fully Assigned.	yes, no	yes
Environment	prehighlight	Yes - Selectable items beneath the pointer are highlighted prior to being selected. No - Selectable items beneath the pointer are not highlighted prior to being selected.	yes, no	
Environment	prehighlight_tree	Default value for "Preselection Highlighting" option in the model tree, layer tree, detail tree and layout tree. When enabled, selectable items are highlighted prior to being selected in the tree.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Environment	pro_unit_length	Sets the default units for new objects.	unit_inch, unit_foot, unit_mm, unit_cm, unit_m	unit_inch
Environment	pro_unit_mass	Sets the default units for mass for new objects.	unit_ounce, unit_pound, unit_ton, unit_gram, unit_kilogram, unit_tonne	unit_pound
Environment	pro_unit_sys	Specifies the default system of units for new models.	mks, cgs, mmns, fps, ips, proe_def, mmks	
Environment	proe_memory_buffer_size	Specify the size in megabytes of the memory buffer to be reserved by Creo Parametric for system out of memory condition. You must restart Creo Parametric for the change to take effect.		50
Environment	regen_backup_directory	Specifies the directory in which the system stores regen_backup models. (See regen_backup_using_disk.) The primary default is the current directory, but if that directory is read-only, the secondary default is the /tmp directory.		
Environment	regen_backup_using_disk	Specifies whether or not the system backs up the current model to disk before each regeneration.	yes, no	no
Environment	save_section_surfaces	Yes - Save section surfaces with models. No - Do not save section surfaces with models.	yes, no	yes
Environment	select_on_dtm_edges	All_modes - Select a datum plane by clicking on its visual boundary. Sketcher_only - Select a datum plane by clicking on its tag in modes other than Sketcher. If you use Query frequently, set this option to Sketcher_only.	all_modes, sketcher_only	all_modes
Environment	show_dim_sign	No - Dimension values always appear positive. Negative value creates geometry to opposite side. Yes - Entering negative value for dim displayed as negative creates geometry to same side. Dims for csys and datum point offsets always show neg/pos value.	yes, no	no
Environment	show_selected_item_id	Show id of Creo Parametric and Sketcher geometry in query selection bin, collector and tool tip.	yes, no	no
Environment	spin_center_display	Determines whether or not the spin center symbol displays.	yes, no	yes
Environment	start_appmgr	Determines whether or not the application manager starts when Creo Parametric is started.	yes, no	yes
Environment	startup_check_for_visualization	Determines whether or not the visualization components are checked for installation upon startup.	yes, no	yes
Features	allow_anatomic_features	Allow the creation of pre-Pro/ENGINEER 2000i geometry features including ear, neck, lip, slot, flange, shaft, local push, radius dome and section dome.	yes, no	no
Features	allow_fully_dependent_copy	Enables the usage of advanced copy/paste functionality to control degree of dependency on the pasted object.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Features	allow_udf_style_cosm_threads	Defines the Cosmetic Thread user interface. Yes - Creates a Cosmetic Thread as a UDF. No - Creates a Cosmetic Thread as a Cosmetic Thread feature.	yes, no	no
Features	angle_grid_interval	Modify grid space in angular units for handle movement.		1
Features	auto_add_remove	Allows Creo Parametric to automatically switch from add to remove of material for solid extrude and revolve during feature creation.	yes, no	yes
Features	autohide_copied_group_af	If set to yes, copying a feature group containing an Annotation feature will automatically hide the Annotation Features in the new group.	yes, no	no
Features	autohide_external_sketch	If set to yes, a Sketch feature is automatically hidden after being used as external sketch for a sketch based feature.	yes, no	yes
Features	autoround_max_n_chains_per_feat	Determines the maximum number of edge chains that the Auto-Round algorithm will attempt to round in each Auto-Round member		20
Features	centimeter_grid_interval	Modify grid space in centimeter units for handle movement.		0.1
Features	copy_set_datum_attr	If set to yes, allow to copy set datum associated with datum features..	yes, no	yes
Features	default_dep_copy_option	Sets Full Dependent Copy as a default copy option.	full_dep, dep_dim	dep_dim
Features	display_draggers	Display Dragger for Edit mode.	yes, no	yes
Features	dynamic_preview	Controls the default state of geometry preview within a feature tool. Attached/Unattached - Dynamic attached/unattached geometry preview will be enabled. No - Geometry preview will not be enabled.	no, attached, unattached	attached
Features	enable_auto_regen	Enable Auto Regeneration for Unified Edit Mode.	yes, no	yes
Features	enable_remove_notifications	Presents a notification dialog when the result of a Remove feature creates a surface with a flipped normal, an intersection with non-adjacent geometry or geom check errors. Setting this option to no will not issue the notification.	yes, no	yes
Features	enable_tapered_extrude	Enables you to add a taper to an extrude feature by displaying the Add taper check box and the taper degree value box on the Extrude tab Options tab. yes - The Add taper check box is displayed. no - The Add taper check box is not displayed.	yes, no	yes
Features	fail_unattached_feature	Handles unattached protrusion and cuts, when retrieving pre-Release 15 models. Yes - When a part is retrieved, the system fails an unattached protrusion or cut. No - When a part is retrieved, the system does not fail an unattached protrusion or cut.	yes, no	no
Features	feature_create_auto_begin	In feature creation, determines whether or not the Define button is automatically activated when you open the dialog box. Yes - Button is activated automatically. No - You select the button.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Features	feature_create_auto_ok	In feature creation, determines whether or not the system activates the dialog box OK button automatically when the last required element is defined. Yes - Button is activated automatically. No - You select the button.	yes, no	no
Features	feature_type_default	It sets the solid or surface default type for feature creation. The Advanced option sets the default type for Extrude and Revolve to solid, and all others to Surface.	solid, advanced, surface	solid
Features	flex_maintain_solution_topology	Initial value for the "Maintain solution topology" option in created flexible modeling features.	yes, no	no
Features	foot_grid_interval	Modify grid space in foot units for handle movement.		0.083333
Features	hole_diameter_override	No- Uses hole-table-driven standard hole diameter value. Yes - Allows specifying the drilled hole value instead of the standard value.	yes, no	no
Features	hole_file_resolution	Specify Hole tool to use external hole parameter (.hol) files.	replace_with_external, use_internal	replace_with_external
Features	hole_parameter_file_path	Specifies the directory path for a standard screw-size parameters (.hol) file to be loaded from disk. Use the full path name to avoid problems.		
Features	inch_grid_interval	Modify grid space in inch units for handle movement.		0.03125
Features	keep_info_datums	Sets the initial state of how the system treats datums and features created at run time.	yes, no	yes
Features	max_regen_time_attach_preview	Sets the maximum regeneration time for the dynamic attached preview.		0.5
Features	merge_type	Sets the default merge type surface merge operations.	join, intersect	intersect
Features	meter_grid_interval	Modify grid space in meter units for handle movement.		0.001
Features	millimeter_grid_interval	Modify grid space in millimeter units for handle movement.		0.1
Features	pro_group_dir	Sets the default directory for the user-defined feature (UDF) library. If not specified, your current working directory is the default. Use the full pathname to avoid problems.		
Features	ratio_grid_interval	Modify grid space in percentage units for handle movement.		0.05
Features	show_geom_checks_on_creation	Yes - The SHOW ERRORS menu appears at the end of feature creation when the feature being created has Geometry Checks.	yes, no	no
Features	troubleshooter_show_info_gcheck	When set to Yes this option will add a "Show info items" filter to the View panel in the Troubleshooter.	yes, no	no
Features	udf_af_name_postfix	If set to yes, a sequential number will be appended to the name of annotation feature in group from Copy and UDF placement.	yes, no	yes
Features	user_defined_grid_interval	Modify grid space in user defined length units for handle movement.		0.5

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
File Storage & Retrieval	analysis crt tmp file on regen	Determines whether regeneration of analysis features creates/updates tmp files	yes, no	yes
File Storage & Retrieval	browser_favorite	Specifies the directories visible in the file browser for quick navigation. Use the full path to avoid problems.		
File Storage & Retrieval	compress_output_files	Yes - Stores object files in a compressed format. Note: Compressed files are usually smaller, but may be larger, than uncompressed files. Compressed files are slower to read and to write to.	yes, no	no
File Storage & Retrieval	dir_cache_max	Specify the maximum number of directories to cache. Larger values can speed up file access, but use more memory.		256
File Storage & Retrieval	disable_search_path_check	Controls whether the search path is checked for name conflicts when creating, renaming, or copying models. Name conflicts should be avoided because different models with the same name cannot be together in session.	yes, no	no
File Storage & Retrieval	excel_analysis_directory	Sets the default directory for the MS Excel files to be used for Excel analysis calculations. Use the full pathname to avoid problems.		
File Storage & Retrieval	excel_family_table_keep_file	Yes - in case of editing family table with excel, if saved excel file exist , will always update it.	yes, no	no
File Storage & Retrieval	exec_prog_on_assem_retr	Execute pro/program for models during assembly retrieval.	yes, no	no
File Storage & Retrieval	external_analysis_directory	Sets the default directory for the files to be used by External Analysis calculations. Use the full pathname to avoid problems.		
File Storage & Retrieval	file_open_default_folder	Sets default dir for File>Open. Working_directory - Search working directory. In_session - Search objects in session. Pro_library - Use only if Pro/LIBRARY is installed. Workspace - Use only with Pro/INTRALINK. Commonsense - Use only with Pro/INTRALINK.	default, working_directory, in_session, pro_library, my_documents, desktop, workspace, server	
File Storage & Retrieval	file_open_default_view	Specify the initial Views setting as List or Details in the File dialog boxes.	list, details	list
File Storage & Retrieval	file_open_preview_default	Controls whether the preview area is expanded by default when the file open dialog or local file browser is displayed.	expanded, collapsed	collapsed
File Storage & Retrieval	file_open_preview_delay	The delay before a file selection is previewed, in tenths of a second.		
File Storage & Retrieval	force_new_file_options_dialog	Force the use of the New File Options dialog.	yes, no	no
File Storage & Retrieval	generate_viewable_on_save	Controls whether updated Creo View .edz files are automatically exported every time a model is saved.	yes, no	no
File Storage & Retrieval	geometry_analysis_color_scheme	Specifies the location of the geometry analysis color scheme file.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
File Storage & Retrieval	instance_search_exhaustive	Specifies extent of instance search. Yes - Include file type when entering instance name in File>Open: type in input panel (e.g. inst_name.prt) or select in drop-down list. If no .idx files are found, only working dir is searched; each generic is not.	yes, no	no
File Storage & Retrieval	lang_propagate	Specifies language storage options. The option provides flexibility for objects that use a different language than the current	yes, no	yes
File Storage & Retrieval	mathcad_analysis_directory	Sets the default directory for the Mathcad files to be used for Mathcad analysis calculations. Use the full pathname to avoid problems.		
File Storage & Retrieval	menu_show_instances	Determines whether instance names listed in instance index files appear in file lists.	yes, no	yes
File Storage & Retrieval	mfg_start_model_dir	Provides the complete path to the directory containing mfg start assemblies.		
File Storage & Retrieval	model_rename_template	Sets the default renaming convention for copied assemblies. The default new name is the old name appended with an underscore (_). Adding a template such as &#91;*a1*="b1*] changes every occurrence of a1 to b1 (e.g. TOP_A1_BRACKET copies to TOP_B1_BRACKET).		
File Storage & Retrieval	modified_objs_in_new_dir	Yes- During Save a Copy operations, any modified object that is not renamed will be saved in the new directory. No- Any modified object that is not renamed will be saved to the retrieval directory.	yes, no	no
File Storage & Retrieval	open_draw_simp_rep_by_default	Set to "Yes" to always invoke the open rep dialog when opening a drawing.	yes, no	no
File Storage & Retrieval	open_simplified_rep_by_default	Retrieves a model in a specified representation. Yes - uses Open Rep dialog. No - retrieves the Master representation. Specified Name - retrieves the model in the specified representation name if it exists or the Default representation.		no
File Storage & Retrieval	override_store_back	Forces the storage of objects retrieved from other directories to the current working directory.	yes, no	no
File Storage & Retrieval	placement_rep_by_default	Retrieves the assembled component in a specified representation. None - retrieves the representation specified in configuration option open_simplified_rep_by_default.		none
File Storage & Retrieval	preferred_save_as_type	Allows customizing the order of the Type selections found under File ->Save A Copy.		
File Storage & Retrieval	pro_crosshatch_dir	Specifies a default directory for your cross-hatch library in which you can save cross-hatching patterns for later retrieval. Its value is the full path name of the default directory.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
File Storage & Retrieval	pro_font_dir	Sets the default font directory (Creo Parametric will load all fonts in this directory into the drawing regardless of the contents of the system and working directories).		<creo3.0_loadpoint>\text\fonts\
File Storage & Retrieval	pro_library_dir	Sets the default directory for the Creo Parametric libraries (or any libraries you have built containing the proper index menu files). Use the full pathname to avoid problems.		
File Storage & Retrieval	pro_material_dir	Sets the default directory for the part material library. Use the full pathname to avoid problems.		
File Storage & Retrieval	pro_surface_finish_dir	Sets the default directory for your user-defined surface finish symbols. Use the full pathname to avoid problems.		<creo3.0_loadpoint>\symbols\suffins\
File Storage & Retrieval	prompt_on_erase_not_disp	Specifies whether or not to display a prompt allowing you to save your undisplayed objects. This option is used in conjunction with the Erase > Not Displayed command on the File menu.	yes, no	no
File Storage & Retrieval	propagate_change_to_parents	When saving objects with option save_objects set to Changed or Changed_and_specified, Creo Parametric verifies which models are. Yes - Any model that is a parent of a changed model is considered to be a changed model when determining what to save.	yes, no	no
File Storage & Retrieval	read_famtab_file_on_retrieve	No - Ignores filename.ptd. Yes - Creates and saves filename.ptd and uses that file when retrieving the generic.	yes, no	no
File Storage & Retrieval	regen_simp_retrieve	Regenerate the assembly when retrieving an assembly simplified representation. Yes - Regenerates the assembly on retrieval. No - The assembly is not regenerated on retrieval.	yes, no	no
File Storage & Retrieval	rename_drawings_with_object	Controls whether the system copies associated drawings automatically with parts and assemblies.	none, both, part, assem	none
File Storage & Retrieval	retain_display_memory	Determines if the display of an object currently on the screen is kept in memory when you quit the window. Doing so (yes) makes the retrieval of objects in memory faster.	yes, no	yes
File Storage & Retrieval	save_clipped_view	Yes - clipping status is saved with model. No - clipping status is not saved.	yes, no	no
File Storage & Retrieval	save_dialog_for_existing_models	Controls whether the Save dialog is shown for models whose storage location is already known.	yes, no	no
File Storage & Retrieval	save_file_iterations	Yes - Iterations of a file will be saved. No - Iterations of a file will not be saved.	yes, no	yes
File Storage & Retrieval	save_instance_accelerator	With family tables of solid parts, determines saving instance accelerators. None - Accelerator files not used. Explicit - Save when instance is explicitly saved. Saved objects - Save when objects were modified. Always - Always save.	none, explicit, always, saved_objects	saved_objects

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
File Storage & Retrieval	save_model_display	Sets amount of graphical data stored. Wireframe - Wireframe data. Shading_high - Most detail (shaded). Shading_low - Least detail (shaded). Shading_lod - Detail determined by View Performance dialog (shaded).	wireframe, shading_low, shading_high, shading_lod	shading_lod
File Storage & Retrieval	save_object_in_current	Use with the override_store_back option. Yes - Objects retrieved from a directory where you do not have write permission will be stored in the current directory. No - These objects are not saved at all.	yes, no	no
File Storage & Retrieval	save_objects	Determines when an object and its dependent objects (such as a part used in an assembly) are stored.	all, changed, changed_and_specified, changed_and_updated	changed_and_specified
File Storage & Retrieval	saving_model_thumbnails	Enables saving of model thumbnails.	yes, no	yes
File Storage & Retrieval	search_path	Specifies list of directories to search (in order) for object/file retrieval. These directories, the working directory, and directories in search.pro file (refer to search_path_file) are Creo Parametric's search path. Use full path name to avoid problems.		
File Storage & Retrieval	search_path_file	Specifies path to text file search.pro (a list of directory path names). The directories listed in search.pro, the working directory, and directories specified by search_path are Creo Parametric's search path. Use full path name to avoid problems.		
File Storage & Retrieval	set_model_types_to_save	Determines which model types are used in the Save Changed command. The format is a list of extensions separated by spaces.		asm prt drw sec mfg gph ipf cem
File Storage & Retrieval	start_model_dir	Provides the complete path to the directory containing start parts and assemblies.		
File Storage & Retrieval	template_boardpart	Specify the model to use as the default new ECAD board template.		
File Storage & Retrieval	template_designasm	Specifies the designated template assembly. Use full path to avoid problems.		inlbs_asm_design.asm
File Storage & Retrieval	template_drawing	Specify the drawing used as the default drawing template.		c_drawing.drw
File Storage & Retrieval	template_ecadasm	Specify the model used as the default ECAD assembly template.		
File Storage & Retrieval	template_ecadpart	Specify the model used as the default ECAD part template.		
File Storage & Retrieval	template_mfgcast	Specify the model used as the default manufacturing cast template.		inlbs_mfg_cast.asm
File Storage & Retrieval	template_mfgcmm	Specify the model used as the default manufacturing cmm template.		inlbs_mfg_cmm.asm

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
File Storage & Retrieval	template_mfgemo	Specify the model used as the default manufacturing expert machinist template.		inlbs_mfg_emo.asm
File Storage & Retrieval	template_mfgmold	Specify the model used as the default manufacturing mold template.		inlbs_mfg_mold.asm
File Storage & Retrieval	template_mfgnc	Specify the model used as the default manufacturing assembly template.		inlbs_mfg_nc.asm
File Storage & Retrieval	template_module	Specifies the default module template file. Use full pathname to avoid problems.		inlbs_asm_module.asm
File Storage & Retrieval	template_mold_layout	Specify the mold layout assembly used as a default template.		inlbs_mold_lay.asm
File Storage & Retrieval	template_new_ecadasm	Specify the model to use as the default new ECAD assembly template.		
File Storage & Retrieval	template_sheetmetalpart	Specify the model used as the default sheetmetal part template.		inlbs_part_sheetmetal.prt
File Storage & Retrieval	template_solidpart	Specify the model used as the default part template.		inlbs_part_solid.prt
File Storage & Retrieval	update_accelerator_in_verify	No* - Does not update existing accelerators upon family table verification for changes. Yes - Update existing accelerators upon family table verification for changes	yes, no	no
File Storage & Retrieval	use_outdated_accel_in_geometry_rep	This option controls the usage of outdated accelerator files during retrieval of instances in geometry representation. Yes - Provides the fastest retrieval. No - Ensures accurate Retrieval.	yes, no	no
File Storage & Retrieval	use_temp_dir_for_inst	Explicitly makes Creo Parametric use the Temp directory for regenerating instances of models.	yes, no	no
File Storage & Retrieval	verify_on_save_by_default	Yes - "Verify Now" action will be selected in the conflict dialog by default when an unverified family table instance is to be saved in PDM workspace. No - "Verify Now" action will not be selected by default. The user may explicitly specify the "Verify Now" action in the conflict dialog.	yes, no	no
Freestyle Feature	freestyle_drag_performance	Yes - Improve performance during drag by temporarily reducing graphics quality. No - No reduction in graphics quality during drag.	yes, no	yes
Layers	add_lower_level_comps_to_layer	Setting this option to yes will check off the Favor Current Layer option, which allows lower level components to be added to top level layers.	yes, no	no
Layers	blank_layer	Blanks specified layers when you begin a Creo Parametric session. The value is the layer ID.		
Layers	create_numbered_layers	Yes - Creates default layers named 1 through 32.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Layers	def_layer	Specifies default layer names for different types of items. The first value string is the layer type. The second value string is the layer name.	layer_assem_member, layer_assy_cut_feat, layer_axis, layer_chamfer_feat, layer_comp_design_model, layer_comp_fixture, layer_comp_workpiece, layer_copy_geom_feat, layer_corn_chamf_feat, layer_cosm_round_feat, layer_cosm_sketch, layer_csys, layer_curve, layer_curve_ent, layer_cut_feat, layer_datum, layer_datum_plane, layer_datum_point, layer_detail_item, layer_dgm_conn_comp, layer_dgm_highway, layer_dgm_rail,	
			layer_dgm_wire, layer_dim, layer_draft_constr, layer_draft_dim, layer_draft_dtm, layer_draft_entity, layer_draft_feat, layer_draft_geom, layer_draft_grp, layer_draft_hidden, layer_draft_others, layer_draft_refdim, layer_driven_dim, layer_dwg_table, layer_ext_copy_geom_feat, layer_feature, layer_geom_feat, layer_gtol, layer_hole_feat,	

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
			layer_intchg_funct, layer_intchg_simp, layer_nogeom_feat, layer_note, layer_parameter_dim, layer_part_refdim, layer_point, layer_protrusion_feat, layer_quilt, layer_refdim, layer_ribbon_feat, layer_rib_feat, layer_round_feat, layer_sfin, layer_shell_feat, layer_skeleton_model, layer_slot_feat, layer_snap_line, layer_solid_geom, layer_surface, layer_symbol, layer_thread_feat, layer_trim_line_feat, layer_weld_feat, layer_all_detail_items, layer_annotation_element, layer_axis_ent, layer_csys_ent, layer_set_datum_tag, layer_3d_items, layer_ae_driving_dim, layer_draft_xsec, layer_hole_thread	
Layers	default_layer_model	Name of the model which is used to drive rule-based layer placement of new items in all models of the same type in session.		
Layers	display_layer	This causes the specified layers to display when you begin a Creo Parametric session. The variable "value" is the layer Id.		
Layers	floating_layer_tree	Set to yes if user wants to see a layer tree dialog separated from navigator.	yes, no	no
Layers	highlight_layer_items	Yes - Items selected in the layer tree are highlighted in the geometry area.	yes, no	yes
Layers	layer_item_highlight_limit	Set the upper limit of items in a layer for highlighting. If items in a layer exceeds this number, highlighting will not occur.		500

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Layers	save_display_status_in_sub_model	If set to no, the top level assembly will store sub model layer display status, allowing lower level display status to be stored without having to save the sub models.	yes, no	yes
Layers	save_hidden_items_w_status	The items on the "Hidden Items" temporary layer are stored permanently when layer display status is saved with Save Status command.	yes, no	yes
Layout	layout_feat_copied_geom_thickness	Layout feature - Copied geometry thickness.	1, 3, 2, 4, 5, 1.5, 2.5, 3.5, 4.5	1.5
Layout	layout_scrollBars	Show/Hides scroll bars.	yes, no	yes
Layout	layout_snap	Enable/Disable snapping to geometry and guides.	yes, no	yes
Layout	layout_symbol_dir	Set and automatically create the default directory for saving and retrieving user-defined symbols. If you do not specify a directory, Layout will use the same directory as Creo Parametric drawings. Use full path name to avoid problems.		<dir_name>\dump_x86_win64\
Manufacturing	allow_mfg_in_assem_mode	Allows a manufacturing model to be opened in assembly mode from the file open dialog. Use "Open In Assembly Mode" option in "File Open" dialog.	yes, no	no
Manufacturing	assy_mfg_open_mode	No Default Application set for opening Assembly NC Models.	mfg, process, feature, any_mode	any_mode
Manufacturing	autodrill_udf_dir	Storage directory for holemaking MUDF's that are used by Pro/NC Auto-drilling functionality		
Manufacturing	autodrilling_config_filename	Retrieve automatically and load the configuration file for the auto-drilling table.		autodrilling
Manufacturing	autodrilling_holestyle_param	Parameter name included in hole features of a manufacturing reference model. The parameter value identifies the MUDF for holemaking that is automatically applied in Pro/NC to drill these holes.		
Manufacturing	campost_dir	Specifies the CAM-Post load point directory to enable you to directly set-up the post-processors and generate MCD files.		
Manufacturing	chk_part_surfs_prof_pock	Yes - Includes all reference part surfaces as check surfaces for profiling and pocketing NC sequences.	yes, no	no
Manufacturing	cl_arrow_scale	Enables you to control the size of the tool path arrow for contouring NC sequences. 0 - The arrow does not display. Any other value - The arrow scales accordingly.		1
Manufacturing	cl_rotate_sign_convention	TOOL or TABLE - defines ROTATE sign direction convention based with respect to either rotation table or cutting tool.	table, tool	table

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Manufacturing	cmm_custom_tool_param_file	Defines the file that contains the user-defined tool parameters' list.		
Manufacturing	display_mfg_icon_for_mfg_assembly	Yes- Display manufacturing icon for manufacturing assembly files in Open/Save dialogs. No- the manufacturing assembly files have assembly icons.	yes, no	no
Manufacturing	enable_2x_traj_multi_loops	Enables selection of loops on surfaces for curve trajectory sequence.	yes, no	no
Manufacturing	enable_param_n_tool_upd_dialog	Enable parameters and tool updation dialog.	yes, no	yes
Manufacturing	gpostpp_dir	Specifies the directory containing the post processor files to be used by Pro/NCPOST (GPOST).		
Manufacturing	ignore_tmpl_pos_for_new_tool	Ignore pocket number from template for tools added to a workcell. Use first available pocket.	yes, no	no
Manufacturing	mfg_auto_ref_mfg_template	Enable/Disable auto referencing for Manufacturing template	yes, no	no
Manufacturing	mfg_auto_ref_prt_as_chk_srf	Yes - In 3, 4, and 5 axis contour and conventional mill sequences, the entire reference part is selected as check surfaces by default when computing the NC sequence toolpath for those sequences. No - You select the check surfaces.	yes, no	yes
Manufacturing	mfg_auto_retract_clear_dist	Specifies distance between automatically created retract plane and step coordinate system. Will be applied to the new steps created in process manager if operation retract cannot be used.		0
Manufacturing	mfg_comments_text_area_size	Controls the number of lines available for comments without scrolling in NC steps dashboard UI.		8
Manufacturing	mfg_custom_tool_param_file	Defines the file that contains the user-defined tool parameters' list.		
Manufacturing	mfg_customize_dialog	Yes - Uses the new customize dialog box for customizing NC sequences. No - Uses the Release 19.0 menus.	yes, no	yes
Manufacturing	mfg_cutline_save_mesh	Yes - Save / Retrieve Mesh for subsequent cutline recalculations, No - New Mesh will be computed for each cutline recalculations.	yes, no	yes
Manufacturing	mfg_ijk_num_digits	Sets the number of digits for i, j, k data points in a CL data file.		10
Manufacturing	mfg_info_location	top left /bottom right -Position of the mfg. Info dialog box.	top_left, bottom_right	top_left
Manufacturing	mfg_modal_command_file	Enables you to make some of the APTn index topics modal, which means that you repeatedly get a list of major keywords corresponding to this topic upon completing a command line.		
Manufacturing	mfg_param_auto_copy_from_tool	Do not copy any parameter of the tool to the sequence parameter tree.	all, none, misc, cutting	none

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Manufacturing	mfg_process_print_dir	Specifies the directory where Mfg Process html page get stored. Use the full pathname to avoid problems.		
Manufacturing	mfg_process_sync_by_default	Specifies whether all steps created in the Process Manager get automatically synchronized with features in the Pro/NC assembly. You can change steps later in the dialog box open during the session.	yes, no	yes
Manufacturing	mfg_process_table_setup_dir	Specifies the directory where Mfg Process Table settings get stored. Use the full pathname to avoid problems.		
Manufacturing	mfg_process_template_dir	Specifies the directory where Mfg Process Templates get stored. Use the full pathname to avoid problems.		
Manufacturing	mfg_setup_dir	Specifies the directory which contains all manufacturing related setup files for the current session.		
Manufacturing	mfg_stock_display_show_default	Sets the default visibility status of the Stock Model after creation.	yes, no	no
Manufacturing	mfg_tbl_comment_from_retrieve_tool	Controls whether the mfg tool comment is getting copied from TOOL_COMMENT parameter of the retrieved tool to the tool manager table comment.	yes, no	no
Manufacturing	mfg_template_dir	Specifies the storage directory for NC templates. Specify full path to avoid problems. NC templates are used in Expert Machinist. Templates store the recipe for a toolpath as the method of tool retrieval.		
Manufacturing	mfg_tool_dialog_setup_file	Specifies the XML based file to setup Tool Dialog.		
Manufacturing	mfg_tool_preview_by_pick	Enables preview of a milling tool at a specified location on the tool path or geometry of the part.	yes, no	yes
Manufacturing	mfg_udf_info_setup_file	Specify name and path of the file containing udf and mfg criteria values .		
Manufacturing	mfg_workpiece_transparency	Controls the workpiece transparency in shaded model display. You can change the value in the range between 0.1 and 1. The bigger the value, the more transparent the workpiece. The default value is 0.5.		0.67
Manufacturing	mfg_xyz_num_digits	Sets the number of digits for x, y, z data points in a CL data file.		10
Manufacturing	nc_autodoc_param_dir	Location of the setup file that describes which process parameters are included in the parameter table.		
Manufacturing	nc_autodoc_report_dir	Location for storing the report and images generated by the shop floor documentation tool.		
Manufacturing	nc_autodoc_setup_dir	Location for documentation setup files that describe which process parameters are included in the parameter table.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Manufacturing	nc_autodoc_template_dir	Location of the style sheet and default images used for shop floor documentation.		
Manufacturing	nccheck_type	Vericut will be used for NC material removal simulation.	vericut, nccheck	vericut
Manufacturing	ncl_file_extension	Sets the default file extension for Pro/MFG CL data files. Enter the file extension only (without the preceding "."). For example, apt produces filename.apt.		ncl
Manufacturing	ncmdl_bar_stock_part_path	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<creo3.0_loadpoint>\text\nncmdl_data\nccmdl_bar.prt
Manufacturing	ncmdl_billet_stock_part_path	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<creo3.0_loadpoint>\text\nncmdl_data\nccmdl_billet.prt
Manufacturing	ncpost_type	Set to GPOST to access the default post processor supplied with Pro/NCPOST.	gpost, ncpost	
Manufacturing	ncseq_outbound_curve	Yes - Create out bound curve.	yes, no	no
Manufacturing	new_turn_scan_type	Allows to use New turn scan types for Turning NC sequences.	yes, no	yes
Manufacturing	pro_mdb_dir	Sets the default directory for the Machinability Database (MDB) files. Use the full pathname to avoid problems.		
Manufacturing	pro_mf_cl_dir	Sets the default directory for the manufacturing CL data files. Use the full pathname to avoid problems.		
Manufacturing	pro_mf_clamp_dir	Specifies the directory where the manufacturing clamps can be found.		
Manufacturing	pro_mf_param_dir	Sets the default directory for the manufacturing parameter files. Use the full pathname to avoid problems.		
Manufacturing	pro_mf_tape_dir	Sets the default directory for the manufacturing TAPE data files. Use the full pathname to avoid problems.		
Manufacturing	pro_mf_tprm_dir	Sets the default directory for the manufacturing tool files. Use the full pathname to avoid problems.		
Manufacturing	pro_mf_workcell_dir	Sets the default directory for the manufacturing workcell files. Use the full pathname to avoid problems.		
Manufacturing	pro_mfg_cmdsyn_dir	Sets the default directory for the syntax menu files in Pro/MFG. Use the full pathname to avoid problems.		
Manufacturing	profile_output_utils	Affects CL-Data output for profile milling sequences. Yes - Sequence parameter output_point is available with options: tip - Toolpath is output with respect to the tool center; on_profile - Toolpath is output with respect to the profile of the tool.	yes, no	no
Manufacturing	proncpst_dbf	Enter directory path to NCPost database file.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Manufacturing	prorembatch_queue_manager	Enables toolpath computation on a separate host machine. Required values are Processor name, Remote machine's hostname, Type of transport (always 0), RPC processor number (same as that of remote processor), Version of queue manager.		
Manufacturing	reverse_spiral_finish_type	If the option value is 1, the tool path is obtained from standard one by reversing it; if the option value is 2, the tool path starts from an inner circle.		
Manufacturing	shade_mfg_ref_models	No - Does not process reference parts when generating the shaded image (to save time). Yes - Shades reference parts when generating shaded images.	yes, no	yes
Manufacturing	store_tool_path_file	Specifies how tool path information is computed. Yes - Tool path information stored in file filename.tph (filename is manufacturing model name). No - Tool path is computed each time manufacturing model is retrieved or parameters are changed.	yes, no	yes
Manufacturing	tool_motion_list_length	Controls the number of rows visible in the tool motion list without scrolling. This value is applied to dashboard based NC Steps.		7
Manufacturing	tool_path_preview_default	Sets tool path preview default mode to OFF.	off, on, auto	auto
Manufacturing	turn_avoid_holder_interference	Yes - Tool holder interference will be ignored during Turning toolpath calculation.	yes, no	no
Manufacturing	wcell_fixt_info_setup_file	Specify name and path of the file containing workcell and fixture name info.		
Manufacturing	workpiece_instances	Cut out are created in Workpiece but not the family table instances.	none, operations, steps, operations_and_steps	none
Mechanism	mdo_cl_executable	Path for Custom Load executable.		
Mechanism	mdo_integration_method	Select integration method for dynamic analyses.	explicit, implicit, auto	auto
Mechanism	mdo_spring_wire_radius	Spring wire radius for dynamics spring icons.		0.15
Mechanism	mdx_belt_icon_width	Width of belt transmission icons.		5
Mechanism	mechanism_significant_digits	The number of significant digits of mantissa for real double values in mechanism mode. Can be set in interval from 6 to 16. Default value is 12.		12
Mechanism	motion_envlp_alert	Displays the Motion Envelope alert the first time the quality level is increased.	yes, no	yes
Mechanism	move_mechanism_in_nongeom_reps	Allow mechanism operations in non geometrical representations.	yes, no, exclude_to_move	yes
Miscellaneous	allow_ply_cross_section	Yes - Enables Pro/COMPOSITE to create a cross-section of composite plys.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Miscellaneous	mapkey	Stores a keyboard macro. Use File > Options > Environment > Mapkey Settings to create, edit and delete mapkeys.		
Miscellaneous	mapkeys_execution	Mapkeys execution behavior. No Feedback - The mapkey is executed with no feedback, Execute with Feedback - A warning message is given upon failure during execution, Stop at failure - The execution stops at failure and a feedback message is provided.	no_feedback, execute_with_feedback, stop_at_failure	no_feedback
Miscellaneous	relation_text_trail_output	No - Disable trail output from Text layout in Relation Parameter UI if it hasn't been changed.	yes, no	yes
Miscellaneous	relation_tool_mapkey_behavior	Will control how mapkeys will be executed in relation tool / dialog. Incremental will add/remove text incrementally to the current set. Full_output will record the original set state and replace the current one with it. Note: this config is dependent on relation_text_trail_output.	increment, full_output	increment
Miscellaneous	show_cav_fit_loc_sys	Allows the printing of the transformation matrix of a Pro/Verify Fit into a file.	yes, no	no
Model Display	allow_solid_perspective_hlr	Yes - Allows a solid to be displayed with hidden lines while in perspective.	yes, no	yes
Model Display	animate_component_placement	Animate the transition between component states in placement.	yes, no	yes
Model Display	animate_explode_states	Animate the transition between explode states and mold opening steps.	yes, no	yes
Model Display	blended_transparency	Yes - Transparent colors will appear using alpha blending (if supported) when the model is shaded.	yes, no, independent	independent
Model Display	capped_clip	Yes - Displays the model as a solid when shaded and clipped. No - Displays the model as surfaces when shaded and clipped.	yes, no	yes
Model Display	clip_always	Yes - Display clipping after Visibilities dialog box is closed.	yes, no	yes
Model Display	datum_point_symbol	Specifies the shape of datum point symbols displayed in Part or Assembly mode.	dot, cross, circle, triangle, square	cross
Model Display	default_ramp_size	Specifies a number of shades of gray to show shading of non-colored objects.		
Model Display	disp_comp_name_in_symb_rep	Displays the names of components in symbolic representations.	yes, no	yes
Model Display	display	Wireframe - All solid edges in white. Hiddenvvis - Hidden lines in grey. Hiddeninvis - Hidden lines removed. Shade - Display shaded surfaces. Settings are reflected in printing and some export modes.	wireframe, hiddenvvis, hiddeninvis, shade, shadewithreflect, shadewithedges	shade
Model Display	display_axes	Displays or hides datum axes.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Model Display	display_axis_tags	Displays or hides datum axis name tags.	yes, no	no
Model Display	display_coord_sys	Yes - Coordinate systems will be displayed. No - Coordinate systems will not be displayed.	yes, no	yes
Model Display	display_coord_sys_tags	Yes - Coordinate system name tags will be displayed. No - Coordinate system name tags will not be displayed.	yes, no	yes
Model Display	display_florin	Yes: display of the florin (international currency symbol) where possible. No: use the euro symbol exclusively.	yes, no	no
Model Display	display_plane_tags	Yes - Datum plane name tags will be displayed. No - Datum plane name tags will not be displayed.	yes, no	no
Model Display	display_planes	Yes - Datum planes will be displayed. No - Datum planes will not be displayed.	yes, no	yes
Model Display	display_point_tags	Yes - Datum point tags will be displayed. No - Datum point tags will not be displayed.	yes, no	no
Model Display	display_points	Yes - Datum points and their names will be displayed. No - Datum points and their names will not be displayed.	yes, no	yes
Model Display	display_silhouette_edges	Sets the display of silhouette edges for wireframe display only.	yes, no	yes
Model Display	display_z_extension_lines	Controls the display of dimension z-extension lines. Yes - z-extension lines will be displayed; No - z-extension lines will not be displayed.	yes, no	yes
Model Display	edge_display_quality	Controls display quality of an edge for wireframe and hidden-line removal. Normal - Normal quality of edge display. High - Increases tessellation by 2. Very_high - Increases tessellation by 3. Low - Decreases tessellation from normal, speeds up display.	high, low, normal, very_high	normal
Model Display	enable_ambient_occlusion	Enable ambient occlusion. This simulates the effect of light radiating from a real model by darkening the occluded areas.	yes, no	no
Model Display	enable_hlr_hashing	Enables hlr hashing	yes, no	no
Model Display	enable_transparent_text_bg	Yes - Clear background before displaying text. No - Do not clear text background.	yes, no	yes
Model Display	explode_animation_max_time	Sets the maximum amount of time for an animation between explode states.		1
Model Display	fast_highlight	Yes - Use simplified highlighting which redisplayes more quickly at the expense of certain details. This option may increase memory usage slightly.	yes, no	yes
Model Display	fasthlr	Specifies whether fast hidden line removal is on/off by default.	yes, no	no
Model Display	frames_per_sec	Determines whether view is animated.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Model Display	frames_per_sec_value	Used to set the smoothness of view animation changes. Higher numbers provide more frames/sec, and giving you a smoother animation. Lower numbers cause the view to change more quickly.		3
Model Display	lods_enabled	Determines whether or not the system uses Level of Detail in shaded models during dynamic orientation (panning, zooming and spinning).	yes, no	no
Model Display	lods_value	When lods_enabled is Yes, this value specifies the default value used to display LODs. The value is a percentage of the edge size vs the total size of the model.		50
Model Display	max_animation_time	When the orientation, or zoom state, of the Creo Parametric window changes, an animation sequence of a predefined duration takes place. This option controls the maximum time the system spends animating.		1
Model Display	max_image_dimension	Set the upper bound dimension on images loaded into Creo Parametric. Images larger than this limit will be rescaled to this upper bound during loading.	4096, 1024, 2048, 8192, 16384	16384
Model Display	mesh_spline_surf	Turns the blue mesh surface lines on or off.	yes, no	no
Model Display	min_animation_steps	When the orientation, or zoom state, of the Creo Parametric window changes, an animation sequence of predefined duration takes place. This option controls the minimum number of frames used in the animation sequence.		6
Model Display	model_detail_options_file	Sets the path to the file which contains the default model detail option values for new models.		<creo3.0_loadpoint>\text\3d_inch.dtl
Model Display	orientation	Establishes the initial standard view orientation. User_default - System uses X-axis and Y-axis values defined by options x_axis and y_axis.	isometric, trimetric, user_default	trimetric
Model Display	real_time_rendering_display	Set the realtime rendering to project shadows/reflections on a transparent floor.	transparent_floor, room_display	transparent_floor
Model Display	save_triangles_flag	Spinning shaded images with software graphics only. Note: This option has no effect if your system uses hardware graphics.	yes, no	no
Model Display	shade_moving_comp	Yes - Shades assembly components while they are being moved. No - Displays components in wireframe while they are being moved.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Model Display	shade_quality	Model shade quality is used to decide how much to subdivide surfaces for shading purposes. Higher shade quality is slower but produces better pictures.	12, 13, 14, 1, 3, 23, 24, 25, 26, 2, 4, 5, 15, 6, 7, 8, 9, 10, 11, 16, 17, 18, 19, 20, 21, 22, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50	3
Model Display	shade_surface_feat	Controls the shading of surface features.	yes, no	yes
Model Display	shade_windows	Enables shading of objects in all windows. This option does not also enable color shading.	all_windows, one_window	all_windows
Model Display	shade_with	Controls the display of datum curves on shaded objects.	no, curves	curves
Model Display	show_axes_for_extr_arcs	Determines if axes are created for newly extruded arcs.	yes, no	no
Model Display	skip_small_surfaces	Gives the user the option not to display small shaded surfaces. Disabling this option will improve display quality at the expense of speed. It is most useful for creating screen snap shots.	yes, no	yes
Model Display	spherical_map_size	Size of Spherical map texture image used for environment mapping. Bigger size will have better image quality but will be slow to regenerate.	256x256, 512x512, 1024x1024	256x256
Model Display	spin_rate_zoom_sensitive	Allows slower spinning when zoomed in on a model.	yes, no	no
Model Display	spin_with_notes	Yes - 3D feature and model notes are displayed during dynamic spinning of a model.	yes, no	yes
Model Display	spin_with_part_entities	Yes - Datum features will be displayed during dynamic spinning of a model.	yes, no	no
Model Display	spin_with_silhouettes	Yes - Silhouette lines will be displayed during dynamic spinning of a model.	yes, no	no
Model Display	tan_angle_for_disp	For display purposes only. Sets the angle between two quilt surfaces below which the edge will be displayed as tangent. Enter an angle between 1.5 and 15 degrees.		0.02618
Model Display	tangent_edge_display	Determines how edges between tangent surfaces are displayed.	no, solid, centerline, phantom, dimmed	solid
Model Display	texture	Yes - Applied textures will be displayed when the model is shaded.	yes, no	yes
Model Display	texture_search_path	Appends a directory to the search path for textures.		
Model Display	transparency	Yes - Transparent colors will appear as such when the model is shaded.	yes, no	yes
Model Display	update_old_appearance	Update pre-Creo 3.0 appearances to their latest values. Always - update without warning, Never - do not update, Prompt - issue a warning dialog allowing users to choose to update or not.	never, always, prompt	always

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Model Display	use_pre_wildfire_text_font	Yes - Use the old stroke-based PTC font as the default font. No - Use a True Type font as the default font.	yes, no	no
Model Display	x_angle	Sets the default view orientation for models (appears in the Orientation dialog box). The default orientation is the last option, x_angle, y_angle, or orientation, in the configuration file.		0
Model Display	y_angle	Sets the default view orientation for models (appears in the Orientation dialog box). The default orientation is the last option, x_angle, y_angle, or orientation, in the configuration file.		0
Notification Manager	nmgr_affected_by_dma	Show notifications for models/features that are affected by Creo direct modifications.	yes, no	yes
Notification Manager	nmgr_dont_show_all	Ignore current configurations and do not display any notifications.	yes, no	no
Notification Manager	nmgr_failed	Show notifications for failed features/components.	yes, no	yes
Notification Manager	nmgr_misssing_mdl	Show notifications for models that could not be reterived.	yes, no	yes
Notification Manager	nmgr_modified_in_dma	Show notifications for models/features that were modified in Creo Direct.	yes, no	yes
Notification Manager	nmgr_outdated_mathcad	Show notifications for models that contain MathCad Sheet with values different from Creo values.	yes, no	yes
Notification Manager	nmgr_outdated_non-native	Show notifications for non-native models that are not up-to-date with respect to source.	yes, no	yes
Notification Manager	nmgr_outdated_pdm	Show notifications for models that are not up-to-date with recent PDM model version.	yes, no	yes
Notification Manager	nmgr_pending_dma	Models that were modified in Creo Direct and not reviewed. The changes should be accepted or rejected in the View Creo Changes dialog	yes, no	yes
Notification Manager	nmgr_reference_changed	Show notifications for models/features whose references were changed since last regeneration.	yes, no	yes
Notification Manager	nmgr_reference_out_of_session	Show notifications for models/features with references not in session.	yes, no	no
Photorender	default_scene_filename	Complete path to the scene file to be used as default scene file.		<creo3.0_loadpoint>\graphic-library\scenes\creo_default_scene.scn
Photorender	photorender_capability_warnings	Allows the user to disable warnings about PhotoLux only capabilities.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Photorender	photorender_default_height	Sets the default height (in pixels) for the custom size rendered output		450
Photorender	photorender_default_width	Sets the default width (in pixels) for the custom size rendered output		600
Photorender	photorender_memory_usage	Sets the memory limit that Pro/PHOTORENDER allows for model processing. Do not use commas in the memory value, as Creo Parametric interprets these as decimals. The minimum allowed value is 64 and maximum value is 5120.		256
Photorender	photorender_preview_scale	Allows user defined size ratio at which Pro/PHOTORENDER will render, using the default Render To: Full Window Preview setting. Increasing the value creates higher quality rendering with slower performance. Reducing the value produces reverse results.		0.5
Photorender	pro_texture_library	Allows the user to specify a different graphics library.		
Photorender	room_rotation_lock	Default room rotation setting at session start. studio - Lock room rotation to studio. model - Lock room rotation to model.	studio, model	studio
Photorender	save_scene_with_file	Yes - Automatically update the model with current scene information when saving the model; No - Scene information embedded in model must be updated manually.	yes, no	no
Photorender	save_texture_with_model	Yes - Embed applied texture and tracesketch bitmap files into the model file.	yes, no	no
Photorender	window_gamma	Gamma correction to apply to images created by renderer. Default value is 1.0.		1
Piping	aec_dwg_anno_attrpar_spec_file	Specifies the aec drawing annotation attribute/parameter specification file name.		aec_dwg_anno_attrparam_spec.ptd
Piping	aec_object_type_file	Specifies the aec object type file name.		aec_object_type.ptd
Piping	display_thick_insulation	Yes - Display insulation in thick mode. No - Display insulation in centerline mode. This option can be overridden by the Thick insulation option in the Model Display dialog box.	yes, no	yes
Piping	display_thick_pipes	Yes - Display pipes in thick mode. No - Display pipes in centerline mode. This option can be overridden by the Thick pipes option in the Model Display dialog box.	yes, no	no
Piping	isogen_attribute_map_file	Specifies the mapping between isogen parameter and attribute name.		<creo3.0_loadpoint>\text\piping_data\isodata\isogen_attribute_map.ptd

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Piping	isogen_endtype_map_file	Specify name of file mapping Pro/Piping endtypes with ISOGEN endtypes		<creo3.0_loadpoint>\text\piping_data\isodata\isogen_endtype_map.ptd
Piping	isogen_mandatory_attr_file	Specify file containing mandatory attribute filenames used by ISOGEN.		isogen.flis
Piping	isogen_nominal_size_map_file	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches		<creo3.0_loadpoint>\text\piping_data\isodata\isogen_nominal_size_map.ptd
Piping	isogen_output_files_dir	Specify directory to save the PCFs created by ISOGEN interface		
Piping	isogen_pcf_filename_format	Specify format to create filenames of PCF files being created		mnemonic,-,specification,-,number
Piping	isogen_symbol_map_file	Specify name of file for mapping Pro/Piping selection names with ISOGEN symbol keys (SKEY)s		<creo3.0_loadpoint>\text\piping_data\isodata\isogen_symbol_map.ptd
Piping	pipe_3d_bend_theor_int_pts	Yes - Displays theoretical intersections and bends in three-dimensional models.	yes, no	no
Piping	pipe_bend_locations_csys_units	Define which assembly units will be used for the piping bend locations calculations. Yes - to use coordinate system assembly units. No - to use pipeline assembly units.	yes, no	no
Piping	pipe_pre_22_bendinfo_conv	Convert bend report information generated prior to Pro/ENGINEER 2000i-2 to current saved analysis format.	yes, no	yes
Piping	pipe_solid_centerline	Yes - Display Pro/PIPING centerlines. No - Do not display Pro/PIPING centerlines.	yes, no	yes
Piping	pipe_solid_label_format	Specify the pipe solid label format that will be used to automatically generate pipeline labels.		size,-,specification,-,mnemonic,-,number,-,insulation
Piping	pipe_update_pre_20_int_pts	Yes - Pipelines created prior to Release 20.0 are updated with intersection point entities. The entities are created in session by Info>Regen Info or when you retrieve the model.	yes, no	no
Piping	pipeline_assembly_library_dir	Sets the default pipeline assembly library directory.		<creo3.0_loadpoint>\text\piping_data\pipelinelib\
Piping	pipeline_assembly_name_format	Sets the default pipeline assembly name format.		mnemonic,-,number

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Piping	pipeline_label_format	Specify the pipeline label format that will be used to automatically generate pipeline labels.		size,-,specification,-,mnemonic,-,number,-,insulation
Piping	pipeline_solid_start_part_name	Sets the default pipeline solid start part name.		pipeline_solid_start.prt
Piping	pipeline_spool_label_format	Specify the pipeline spool label format that will be used to generate the spool labels during pipeline spool generation.		mnemonic-number-prefix&#91;0]spoolnum&#91;01]suffix&#91;]
Piping	pipeline_start_assembly_name	Sets the default pipeline start assembly name.		pipeline_start.asm
Piping	piping_appearance_map_file	Sets the default piping appearance map filename.		<creo3.0_loadpoint>\text\piping_data\master_catalog\piping_appearance.map
Piping	piping_bolt_nut_select_file	Piping bolts and nuts selection file.		<creo3.0_loadpoint>\text\piping_data\master_catalog\piping_bolt_nut_select.ptd
Piping	piping_design_method	Sets the current Piping design method for pipeline modeling.	non_spec_driven, spec_driven, user_driven	non_spec_driven
Piping	piping_enable_designate_report	Enable designatable reports for spec-driven piping.	yes, no	yes
Piping	piping_end_compatibility_file	Sets the default piping end compatibility filename.		<creo3.0_loadpoint>\text\piping_data\master_catalog\piping_end_compatibility.ptd
Piping	piping_extend_dim_scheme	Specify a dimension scheme for piping extend segment.	cartesian, cylindrical, spherical	cartesian
Piping	piping_extend_show_csys	Control whether small preview CSYS is shown at end of pipe when defining extend features in cartesian / spherical / cylindrical schemes.	yes, no	no
Piping	piping_fitt_angle_tolerance	Sets the default piping fitting angle tolerance value.		1.5
Piping	piping_fitt_category_map_file	Sets the default piping fitting category map filename.		<creo3.0_loadpoint>\text\piping_data\sample_project\piping_fitt_category_map.ptd

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Piping	pipng_fitt_library_dir	Sets the default piping fitting library directory.		<creo3.0_loadpoint>\text \pipng_data\fittinglib\
Piping	pipng_fluid_parameter	Enables/disables the Parameters tab in Create Pipeline Dialog	yes, no	no
Piping	pipng_insulation_dir_file	Sets the default piping insulation directory filename.		<creo3.0_loadpoint>\text \pipng_data\sample_pro ject\pipng_insulation_dir .ptd
Piping	pipng_joint_fitting_clearance	Sets the piping joint minimum fitting clearance value.		1
Piping	pipng_joint_fitting_offset	Sets the piping joint minimum fitting offset value.		1
Piping	pipng_manufacture_dir_file	Sets the default piping manufacture directory filename.		<creo3.0_loadpoint>\text \pipng_data\master_cat alog\pipng_manufacture _dir.ptd
Piping	pipng_material_file	Sets the default piping material filename.		<creo3.0_loadpoint>\text \pipng_data\master_cat alog\pipng_material.ptd
Piping	pipng_material_source	Sets the source for material information. Note Material_dir requires pro_material_dir to be set.	weight_length, material_dir	weight_length
Piping	pipng_mcat_dir	Sets the default piping master catalog directory.		<creo3.0_loadpoint>\text \pipng_data\master_cat alog\
Piping	pipng_mcat_dir_file	Sets the default piping master catalog directory filename.		<creo3.0_loadpoint>\text \pipng_data\master_cat alog\pipng_mcat_dir.ptd
Piping	pipng_min_bolting_clearance	Sets the default piping minimum bolting clearance value.		1
Piping	pipng_project_data_dir	Sets the default Project data directory.		<creo3.0_loadpoint>\text \pipng_data\sample_pro ject\
Piping	pipng_schematic_driven	Set the Schematic Driven Pipeline Modeling ON/OFF	yes, no	no
Piping	pipng_schematic_xml_dir	Specify directory with full path containing schematic information XML file(s).		<dir_name>\dump_x86e _win64\

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Piping	pipng_spec_dir_file	Sets the default piping specification directory filename.		<creo3.0_loadpoint>\text\pipng_data\sample_project\pipng_spec_dir.ptd
Piping	pipng_system_tree_format	Sets the default piping model tree format.		mnemonic
Piping	pipng_thumb_wheel_increment	Sets the default piping thumb wheel increment value.		1
Piping	pipng_thumb_wheel_ratio_increment	Sets the default piping thumb wheel ratio increment value.		0.01
Piping	pipng_wildfire_convert	Always ask before converting old piping assemblies to update the network structure.	never, automatic, manual	manual
Piping	pro_insulation_dir	Specifies the directory to search for insulation files used in Pro/PIPING. Use full path name to avoid problems.		
Piping	pro_pip_fitt_dir	Specifies the directory to search for fittings used in Pro/PIPING.		
Piping	pro_pip_lnstk_dir	Specifies the directory to search for linestock files used in Pro/PIPING. Use full path name to avoid problems.		
Piping	units_system_dic_file	Specifies the file containing the mapping for MCAT filename and units.		<creo3.0_loadpoint>\text\pipng_data\master_catalog\pipng_units_system_dic_file.ptd
Printing & Plotting	acrobat_reader	Sets the Adobe Acrobat Reader command path. This value will be used to launch the reader after exporting a PDF document.		
Printing & Plotting	delete_after_plotting	Yes - A plot file is automatically deleted from the directory in which it resides when it has been successfully plotted.	yes, no	
Printing & Plotting	din_std_line_priority	If set to YES line priority in plotting or export is determined according to DIN standard	yes, no	no
Printing & Plotting	gerber_360_interpolation	Yes - Arcs and circles included in outputted Gerber plot files will take this format.	yes, no	no
Printing & Plotting	pen_slew	Sets the pen speed for both x and y directions for plotters that are compatible with this option. Consult your plotter's user guide for more information.		-1.000000 0.000000
Printing & Plotting	pen_slew_xy	Sets the pen speed for the x and y directions individually for plotters that are compatible with this option. The first value is the x value and the second, the y value. They should be separated by a space.		-1.000000 0.000000
Printing & Plotting	pen_table_file	Specifies a default pen mapping table which supersedes other pen mapping information.		table.pnt
Printing & Plotting	pen1_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		4

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Printing & Plotting	pen2_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		1
Printing & Plotting	pen3_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		2
Printing & Plotting	pen4_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		3
Printing & Plotting	pen5_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		2
Printing & Plotting	pen6_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		3
Printing & Plotting	pen7_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		1
Printing & Plotting	pen8_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		4
Printing & Plotting	plot_carberry_extends	When plotting to a raster format (jpg, tiff, etc), set to YES to ignore the view border and produce output sized to the smallest rectangle that includes the drawing items.	yes, no	no
Printing & Plotting	plot_file_dir	Specifies the directory where plot files should be written. Use the full pathname to avoid problems. For example /home/users/plotfiles.		
Printing & Plotting	plot_linestyle_scale	Specifies the scaling factor for the DOTFONT line style in a plot.		1
Printing & Plotting	plot_names	Yes - Plot files are given descriptive extensions when created. No - All plot files are simply given the extension .plt.	yes, no	
Printing & Plotting	plot_proceed_dialog	Yes - Enables the Plot Proceed dialog which provides a choice for delaying the printing of plot files. This is useful for large plots when using lp print commands. The lp command may not access the plot file before it is removed resulting in no output.	yes, no	no
Printing & Plotting	plot_to_scale_full_window	Provides the ability to exclude empty window space when plotting the model to scale.	yes, no	
Printing & Plotting	plotter	Establishes the default plotter for making plot files.		
Printing & Plotting	plotter_command	Sets the command that you use to start a plot on your system; for Windows NT or Windows 95, windows_print_manager option configures Creo Parametric to plot to a recognized device in the Windows NT or Windows 95 print manager.	windows_print_manager	
Printing & Plotting	plotter_handshake	Specifies the type of plotter handshake initialization sequence generated in the plotter files.	hardware, software	
Printing & Plotting	postscript_compression	Specifies whether or not to compress exported shaded PostScript files. Yes - A shaded postscript file created with File>Export will be compressed. This will increase processing time.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Printing & Plotting	pro_plot_config_dir	Sets the directory of your user-defined plotter configuration file. Use the full pathname to avoid problems.		
Printing & Plotting	ps_output_with_pagesize	Specifies whether or not to include pagesize information when generating postscript files from Creo Parametric. No - Creo Parametric will omit Adobe level 2 pagesize information from the header of the postscript file.	yes, no	yes
Printing & Plotting	raster_plot_dpi	Determines the resolution for plotter outputs variables are in dots per inch.	100, 200, 300, 400	100
Printing & Plotting	rotate_postscript_print	Specifies rotation for printing. Yes - Rotates a PostScript plot 90 degrees counterclockwise. Use when you are plotting a landscape drawing on a portrait-type laser printer, or a portrait drawing on a landscape-type laser printer.	yes, no	
Printing & Plotting	shaded_postscript_background	Specifies how background colors are handled in Postscript files of shaded objects.	yes, no	no
Printing & Plotting	ttf_handling_for_plot_files	Defines how annotations using True Type Fonts are written to the plot file. Using True Type Fonts reduces the plot file size, but font substitution may occur due to device settings. Annotations that do not use True Type Fonts are unaffected by this option.	stroke all fonts, use true type fonts	use true type fonts
Printing & Plotting	u3d_create_color_surfaces	YES- Exports color of each surface of model. NO- Exports one color per model. Default NO.	yes, no	no
Printing & Plotting	use_8_plotter_pens	Specifies whether to support up to 8 plotter pens. The initial default is 4 pens.	yes, no	no
Printing & Plotting	use_software_linefonts	Yes - The exact line style used in Creo Parametric is plotted, stroked out dot by dot, dash by dash, and space by space. No - plots lines using the line font it supports which most closely resembles the font used in Creo Parametric.	yes, no	no
Printing & Plotting	variable_plots_in_inches	No - Variable plot sizes may be entered in millimeters.	yes, no	yes
Printing & Plotting	versatec_cutter_installed	Yes - Indicates that a cutter is installed on a Versatec plotter.	yes, no	
Reference Control	allow_direct_ref_to_layout_file	Allow direct references to layout files through layout feature.	yes, no	no
Reference Control	allow_ref_scope_change	No - Displays the message "Reference scope changes are prohibited by the configuration file settings" when the Ref Scope user interface is changed.	yes, no	yes
Reference Control	def_comp_ref_ctrl_exception	Default setting for newly created components that allows/disallows referencing entities forbidden by Reference Control settings.	yes, no	yes
Reference Control	def_multiple_backup_prompt_num	Number of selected components that will invoke confirmation before multi-component backing up placement references.		100

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Reference Control	def_obj_feat_refs_to_backup	Default setting that controls the level of feature backup for newly created models. Other_models: all references to other models will be backed up, Non_permitted: only references that were created despite the initial restrictions will be backed up.	none, other_models, non_permitted	non_permitted
Reference Control	def_obj_place_refs_to_backup	Default setting that controls the level of placement backup for newly created models. Other_models: all references to other models will be backed up, Non_permitted: only references that were created despite the initial restrictions will be backed up.	all, none, other_models, non_permitted	non_permitted
Reference Control	def_obj_ref_ctrl_exception	Default setting for newly created models that allows/disallows referencing entities forbidden by Reference Control settings.	yes, no	yes
Reference Control	default_comp_geom_scope	Set the specified Shared Geometry reference control conditions to newly added components.	all, none, pg	all
Reference Control	default_comp_module_geom_scope	Set the specified Shared Geometry reference control conditions to newly added components for configurable structure.	all, none, pg, tagged	tagged
Reference Control	default_comp_scope_setting	Set the specified Accessible reference control conditions to newly added components.	all, none, subassemblies, skeleton_model	all
Reference Control	default_comp_skel_geom_scope	Set the specified Shared Geometry reference control conditions to newly added Skeleton components.	all, none, pg	all
Reference Control	default_comp_skel_scope_setting	Set the specified Accessible reference control conditions to newly added Skeleton components.	all, none, subassemblies, skeleton_model	all
Reference Control	default_ext_ref_scope	Set default scope for externally referenced models. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all, none, subassemblies, skeleton_model	all
Reference Control	default_geom_scope	Default value for Geometry Scope allowed for referencing.	all, pg_if_exists, pg_only	all
Reference Control	default_obj_constraints_scope	Set the specified Shared Placement reference control conditions to newly created models if template models were not used.	all, none, interfaces	all
Reference Control	default_obj_module_constraint_scope	Set the specified Shared Placement reference control conditions to newly created models for configurable structure if template models were not used.	all, none, interfaces, tagged	tagged
Reference Control	default_obj_module_geom_scope	Set the specified Shared Geometry reference control conditions to newly created models for configurable structure modules if template models were not used.	all, none, pg, tagged	all
Reference Control	default_object_geom_scope	Set the specified Shared Geometry reference control conditions to newly created models if template models were not used.	all, none, pg	all

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Reference Control	default_object_scope_setting	Set default condition for reference control. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all, none, subassemblies, skeleton_model	all
Reference Control	default_placement_scope	Default value for Component Placement Reference filter.	all, intrf_if_exists, intrf_only	all
Reference Control	feat_references_to_backup	Level of backup for feature references	none, other_models, non_permitted	non_permitted
Reference Control	ignore_all_ref_scope_settings	Controls whether object-specific reference scope settings are ignored or not. Environment scope settings will still be enforced. See also allow_ref_scope_change and default_ext_ref_scope	yes, no	no
Reference Control	ignore_export_geom_declaration	Ignore Export Geometry settings.	yes, no	no
Reference Control	model_allow_ref_scope_change	Yes - Users can change the scope setting of components.	yes, no	yes
Reference Control	placement_references_to_backup	Level of backup for placement references	all, none, other_models, non_permitted	non_permitted
Reference Control	ref_control_allow_exceptions	Check to allow referencing components that are not permitted by the above setting	yes, no	yes
Reference Control	ref_scope_copy_color	Set the color of backed up references according to RGB values. Enter RGB values between 0 and 100 separated by space.		55.000000 55.000000 55.000000
Reference Control	ref_scope_copy_color_change	Enable changing colors of backed up references during selection.	yes, no	yes
Reference Control	ref_scope_no_pick_to_copy	Disable selecting references backed up by references control.	yes, no	no
Reference Control	ref_scope_no_pick_to_prohibit	Disable selecting references prohibited by references control.	yes, no	yes
Reference Control	ref_scope_prohibit_color	Set the color of prohibited references according to RGB values. Enter RGB values between 0 and 100 separated by space.		45.000000 45.000000 45.000000
Reference Control	ref_scope_prohibit_color_change	Enable changing colors of prohibited references during selection.	yes, no	yes
Sheetmetal	close_corner_relief	Closes the gap between the bend surfaces of a corner that is a result of intersecting walls in the form state.	yes, no	no
Sheetmetal	die_models_directory	Sets path to directory that contains the die models.		
Sheetmetal	enable_flat_state	Enables the creation of a flat state instance of the model. The instance can be fully flat or fully formed.	yes, no	no
Sheetmetal	flange_shape_sketches_directory	Sets path to directory that contains the Flange shapes.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Sheetmetal	flat_shape_sketches_directory	Sets path to directory that contains the Flat shapes.		
Sheetmetal	initial_bend_y_factor	Specifies a constant used to determine the neutral bend line for a sheet metal part. This value is always used for non-cylindrical bends. It is used for regular bends only when no bend table is specified.		0.5
Sheetmetal	material_upd_smt_bend_table	Defines whether assignment of material to the model via Creo Toolkit will overwrite existing bend table data and y factor value. Always Replace - Overwrites existing data. Never Replace - Keeps existing data	always_replace, never_replace	always_replace
Sheetmetal	merge_smt_sfs_without_seam	Yes - no edge left at the merge if the sheetmetal surfaces being merged are same surfaces. No - edge is left even between same sheetmetal surfaces.	yes, no	yes
Sheetmetal	pro_sheet_met_dir	Sets the default directory of your user-defined bend tables. If not set, the Pro/SHEETMETAL supplied bend tables are used. Use the full path name to avoid problems.		
Sheetmetal	pro_smt_params_dir	Specifies the directory to save/retrieve Sheet metal parameters files. Use full path name to avoid problems.		
Sheetmetal	punch_axis_points	Yes - Enables the creation of punch axis points in Sheetmetal cuts and punches.	yes, no	no
Sheetmetal	punch_models_directory	Sets path to directory that contains the punch models.		
Sheetmetal	smt_add_bends_on_sharp_edges	Determines if bends will be added automatically on edges with sharp geometry in the wall creation tools. Yes - bends will be added automatically. No - sharp geometry of edges will be kept.	yes, no	yes
Sheetmetal	smt_allow_flip_sketch	Yes - Allows flipping the sketch in Flat and Flange tools. No - Disables the ability to flip the sketch in Flat and Flange tools.	yes, no	no
Sheetmetal	smt_bend_notes_dflt_display	Define the default state of bend note display (sheetmetal mode)	yes, no	yes
Sheetmetal	smt_bend_notes_direction_down	Define the down direction symbol for bend notes (sheetmetal mode)		default
Sheetmetal	smt_bend_notes_direction_up	Define the up direction symbol for bend notes (sheetmetal mode)		default
Sheetmetal	smt_bend_notes_order	Define the fields order for bend notes (sheetmetal mode)		&type&direction&angle
Sheetmetal	smt_bend_notes_type_formed	Define the formed symbol for bend notes (sheetmetal mode)		default
Sheetmetal	smt_bend_notes_type_rolled	Define the rolled symbol for bend notes (sheetmetal mode)		default

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Sheetmetal	smt_crn_rel_display	Yes - Corner relief notes will be displayed. No - Corner relief notes will not be displayed.	yes, no	yes
Sheetmetal	smt_drive_bend_by_parameters	Sheetmetal bend radius and bend angle values and bend dimension location will be driven by feature level relations to Sheetmetal parameters.	yes, no	no
Sheetmetal	smt_drive_tools_by_parameters	Drives the Sheetmetal options and values by feature-level relations to Sheetmetal parameters.	yes, no	yes
Sheetmetal	smt_form_abort_on_fail_plc	Yes - will always fail the Sheetmetal Form feature if there is a problem with its placement .	yes, no	no
Sheetmetal	smt_mp_method	MASS - Suppressed Flat Pattern and Flatten Forms will be temporary resumed before mass properties calculation of sheetmetal part. CG - Mass Properties calculation will be performed on current state of sheetmetal part. BOTH - Both methods will be used one after the other.	both, mass, cg	cg
Sheetmetal	smt_outside_mold_lines	Yes - Outside Mold Lines will be created during Flat Pattern creation.	yes, no	no
Sketcher	2d_palette_path	Sets the default directory for the user sketcher shape library. Use the full path name to avoid problems.		
Sketcher	dma_sketcher_guide_width_factor	Guide width as a proportion of sketched line width		0.75
Sketcher	grid_snap	Yes - Makes pick points snap to a grid. No - Turns the grid snap off so that any location can be picked.	yes, no	no
Sketcher	section_color	Specifies the color of sketched sections.	default, drawing_color, sketch_color	
Sketcher	sket_import_geometry_line_style	Determine if the original line style and color should be preserved on Copy/Paste, Sketcher Palette and Import from file. Yes - Preserve original line style and color.	yes, no	no
Sketcher	sketcher_alignment_constr	Yes - Alignment constraints will be used by Intent Manager; No - Alignment constraints will not be used.	yes, no	no
Sketcher	sketcher_animated_modify	No - The section is not animated as modifications are regenerated.	yes, no	yes
Sketcher	sketcher_auto_create_references	This option controls the automatic reference creation from selected background geometry.	yes, no	no
Sketcher	sketcher_auto_create_refs	The option values are: 2: the system automatically creates 2 dimensioning references; 1: the system automatically adds the orientation reference as a dimensioning reference; 0: the system does not automatically create dimensioning references.	1, 2	2

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Sketcher	sketcher_blended_background	This option controls whether the blended background should be used in 3D Sketcher. The option has no effect if blended background is turned off generally.	yes, no	no
Sketcher	sketcher_collinear_constr	Yes - collinear constraints will be used by Intent Manager; No - collinear constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_default_font_kerning	Automatically set kerning for text entities in Sketcher. Yes - Kerning will be set for new text entities. No - Kerning will not be set automatically.	yes, no	no
Sketcher	sketcher_diagonal_constr	Yes - Diagonal constraints will be used; No - Diagonal constraints will not be used.	yes, no	no
Sketcher	sketcher_dim_of_revolve_axes	If this option is set all dimensions created by Intent Manager to Axis of Revolution will be diameter dimensions.	yes, no	yes
Sketcher	sketcher_dimension_autolock	Automatically lock strong sketcher dimensions.	yes, no	no
Sketcher	sketcher_disp_constraints	Shows the constraints when sketching in Sketcher Mode; for example, H for horizontal, V for vertical, and so on.	yes, no	yes
Sketcher	sketcher_disp_dimensions	No - Suppresses the display of all dimensions while in sketcher.	yes, no	yes
Sketcher	sketcher_disp_grid	Specify whether or not to display the sketcher grid.	yes, no	no
Sketcher	sketcher_disp_guides	Shows guides when sketching.	yes, no	yes
Sketcher	sketcher_disp_vertices	No - Suppresses the display of yellow points on all vertices while in sketcher.	yes, no	yes
Sketcher	sketcher_disp_weak_dimensions	Yes - weak dimensions will be displayed; No - weak dimensions will not be displayed.	yes, no	yes
Sketcher	sketcher_equal_length_constraint	Yes - equal length constraints will be used by Intent Manager; No - equal length constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_equal_radii_constr	Yes - equal radii constraints will be used by Intent Manager; No - equal radii constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_grid_angle	Enter a grid angle value to override the default grid angle value.		0
Sketcher	sketcher_grid_line_quantity	Enter number of radial lines for radial grid.		12
Sketcher	sketcher_grid_radius_spacing	Enter a radial grid spacing value to override the default radial grid spacing value.		1
Sketcher	sketcher_grid_type	Set grid type to be Cartesian or Polar.	cartesian, polar	cartesian
Sketcher	sketcher_highlight_open_ends	Yes - Highlights open ends of sketched entities in Sketcher. No - Does not highlight open ends of sketched entities in Sketcher.	yes, no	yes
Sketcher	sketcher_import_exact_geometry	During import in sketcher use exact geometry.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Sketcher	sketcher_known_dim_on_ref_ent	YES - create known dimensions on known geometry, NO - create reference dimensions on known geometry	yes, no	yes
Sketcher	sketcher_lineup_hor_constr	Yes - line up horizontal constraints will be used by Intent Manager; No - line up horizontal constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_lineup_ver_constr	Yes - line up vertical constraints will be used by Intent Manager; No - line up vertical constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_lock_modified_dims	Yes - modified dimensions will be locked; No - modified dimensions will not be locked.	yes, no	no
Sketcher	sketcher_midpoint_constr	Yes - midpoint constraints will be used by Intent Manager; No - midpoint constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_parallel_constr	Yes - parallel constraints will be used by Intent Manager; No - parallel constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_perpendicular_constr	Yes - perpendicular constraints will be used by Intent Manager; No - perpendicular constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_point_on_entity_constr	Yes - point on entity constraints will be used by Intent Manager; No - point on entity constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_refit_after_dim_modify	Refits section after dimension modification in 2D section or when creating the first feature.	yes, no	yes
Sketcher	sketcher_same_point_constr	Yes - same points constraints will be used by Intent Manager; No - same points constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_save_preview_image	This option controls whether section files should be saved with embedded image information which can be used to preview sections in File/Open dialog.	yes, no	no
Sketcher	sketcher_set_grid_method	"Dynamic" - grid spacing is determined by the system based on zoom factor, "Static" - grid spacing is fixed and set by the user.	dynamic, static	dynamic
Sketcher	sketcher_set_grid_x_major	Enter the number of grid lines per major line in x axis.		5
Sketcher	sketcher_set_grid_x_spacing	Enter an x-grid spacing value to override the default x-grid spacing value.		1
Sketcher	sketcher_set_grid_y_major	Enter the number of grid lines per major line in y axis.		5
Sketcher	sketcher_set_grid_y_spacing	Enter a y-grid spacing value to override the default y-grid spacing value.		1
Sketcher	sketcher_shade_closed_loops	yes-closed loops in sketcher will be displayed as shaded, no-closed loops in sketcher will not be displayed as shaded	yes, no	yes
Sketcher	sketcher_snap_sensitivity	Adjusts the sensitivity of the snapping to geometry.	high, medium, low, very_high, very_low	very_high

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Sketcher	sketcher_starts_in_2d	Defines initial model orientation in Sketcher mode. Yes - (2D orientation) Looking directly at section (sketching) plane. No - (Orientation unchanged) Sketch directly on the 3D part.	yes, no	no
Sketcher	sketcher_symmetric_constr	Yes - symmetric constraints will be used by Intent Manager; No - symmetric constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_tangent_constr	Yes - tangent constraints will be used by Intent Manager; No - tangent constraints will not be used by Intent Manager.	yes, no	yes
Sketcher	sketcher_undo_reorient_view	Controls the ability to Undo view reorientation while in Sketcher. Yes - it is possible to Undo view reorientation in Sketcher.	yes, no	no
Sketcher	sketcher_undo_stack_limit	Sketcher saves a copy of each function performed. The number of possible saved functions depend on the number specified in the option. The undo menu can be used to remove the stored functions.		200
Style Feature	advanced_style_surface_edit	Use advanced options by default for Surface Edit in Style.	yes, no	no
Style Feature	keep_style_items_hidden_on_exit	Keep style entities hidden after exiting of the Style feature definition.	yes, no	no
Style Feature	style_auto_regen_curves	Yes - Enable autoregeneration of curves.	yes, no	yes
Style Feature	style_auto_regen_shaded_surface	Yes - Enable autoregeneration of shaded surfaces.	yes, no	yes
Style Feature	style_auto_regen_surface	Yes - Enable autoregeneration of surfaces.	yes, no	yes
Style Feature	style_connection_arrow_scale	Set scale factor of surface connection arrow. Number should be greater than 0.		1
Style Feature	style_default_connections	Yes - Add connections to a newly created Style surface.	yes, no	yes
Style Feature	style_display_grid	Yes - Display grid for active datum plane.	yes, no	yes
Style Feature	style_grid_spacing	Set the number of lines to display on the grid. Number should be equal or greater than 2.		5
Style Feature	style_merge_surfaces	Merge surfaces on exit from the Style Feature	yes, no	yes
Style Feature	style_surface_mesh_display	On - Display surfaces mesh. Off - Do not display surfaces mesh. Off When Shaded - Display surfaces mesh except when shaded.	off, on, off when shaded	off when shaded
Style Feature	style_surface_mesh_quality	Sets the quality of the surface mesh. Values range is between 0.1 to 10.		1
System	allow_float_opt_checkout	Yes - Allows checkout failure for floating modules.	yes, no	no
System	auto_traceback	"always": a traceback file will be generated when a fatal error occurs. "yes", "ask": ask the user whether to generate a traceback. "no", "never": no traceback will be generated.	yes, no, never, ask, always	always
System	browser_limit_to_ie9_mode	Limit Internet Explorer to IE9 standard mode even if higher version is installed.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
System	cmdmgr_trail_output	Trail file and mapkey output will include system instructions that are independent of user interface configuration when available (typically menu bar and toolbar options).	yes, no	yes
System	cpus_to_use	Set up the number of threads including main thread		2
System	csv_format_delimiter	Define the delimiter character in BOM format file: Comma, TAB, Semicolon, Space or other user-defined	comma, tab, semicolon, space	
System	display_ie8_warning_dialog	Show/Hide the warning dialog related to Internet Explorer 8 running in Compatibility Mode	yes, no	yes
System	enable_tree_indep	Setup whether tree customization will allow separating the tree window from Creo Parametric window.	yes, no	no
System	export_report_format	Controls the format of the exported reports displayed in the embedded browser. Rich Text - Exports the report in Rich Text format, Comma Delimited - Exports the report in Comma Delimited format.	comma_delimited, rich_text	comma_delimited
System	general_undo_stack_limit	Setup the default number of entries in the top level Undo/Redo (default = 50).		50
System	overlays_enabled	Yes - Enables Creo Parametric screens and menus to be placed on different overlay layers of the hardware graphics card. This frees memory for Creo Parametric use.	yes, no	no
System	pick_aperture_radius	Specifies the size of the area about the mouse when making selections. Units are 1/1000 of screen size.		7
System	protable_in_background	Yes - Starts Pro/TABLE editor in the background. Reduces time between pressing a menu button that uses Pro/TABLE and entering the data.	yes, no	yes
System	saveunders_enabled	Yes - Enables Creo Parametric popup helptext and menus to retain the portion of the window they obscure to be used to restore the window image when they are dismissed.	yes, no	yes
System	set_trail_single_step	Yes - Enables a trail file to be single-stepped by entering <CR>.	yes, no	no
System	tablet_device_name	Specifies the name of a digitizing tablet.		
System	trail_delay	Sets a delay in seconds between trail file steps.		
System	trail_dir	Instructs Creo Parametric to create its trail file in the specified directory rather than in the startup directory.		
System	train_file_path	Specifies the default directory for training files. The path name must start from the root of the directory, for example, /home/users/library/training.		
System	web_browser_history_days	Enter number of days to store history records.		20

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
System	web_browser_homepage	Enter location of Creo Parametric browser home page.		<creo3.0_loadpoint>\help\usascii\resource_center_index.html
System	web_browser_in_separate_window	Specifies whether Creo Parametrics web browser should be launched as a separate window. When set to Yes the browser will be displayed as a separate window. The default, No, will result in the browser being embedded in the Creo Parametric application window	yes, no	no
System	windows_browser_type	Select the type of embedded browser to use.	ie_browser	ie_browser
Update Control	child_of_layout_backup_update	Set the default update state for a child referencing the layout feature's geometry.	manual, auto, notify	auto
Update Control	copied_feat_refs_update_state	Copied Feature References Update	manual, auto, notify, no_dependency	auto
Update Control	copied_place_refs_update_state	Copied Placement References Update	manual, auto, notify, no_dependency	auto
Update Control	direct_child_of_layout_update	Set the default update state for a child directly referencing the layout model through layout feature.	manual, auto, notify	notify
Update Control	individual_child_update	Allow individual update control for children of layout feature.	yes, no	yes
Update Control	new_cg_update_state	Set the default update state for newly created Copy Geometry features relative to the reference model.	manual, auto, notify	auto
Update Control	new_layout_feature_update_state	Define the default update state of a new DSF created.	manual, auto, notify	notify
Update Control	retrieve_data_sharing_ref_parts	Automatic reference parts retrieval for dependent data sharing features.	yes, no, ignore_missing	no
User Interface	ae_propagate_detail_dependency	Sets the AE detail dependency for AE's that are locally propagated. If set to independent, the AE's parameter call outs will update to other propagated AEs parameters.	dependent, independent	dependent
User Interface	af_copy_references_flag	Toggle display of Copy References column in Annotation Feature definition dialog	yes, no	no
User Interface	allow_confirm_window	Display a confirmation window when exiting Creo Parametric. This enables you to use the mouse to confirm/cancel exiting Creo Parametric.	yes, no	yes
User Interface	auto_ae_param_file	Define path and file name of external file used to define parameters that are automatically created in all Annotation Elements.		
User Interface	auto_include_annot_plane	Control the default setting of the check box in the Propagate annotation dialog box which controls whether to pass all annotations plane references.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
User Interface	auto_propagate_ae	Yes - causes the automatic, local propagation of AE's after a supported feature has been created.	yes, no	no
User Interface	auxapp_popup_menu_info	Enable display of popup menu info in the trail file.	yes, no	no
User Interface	bm_graph_tool	BMX graph tool	default, excel_linked, excel_embedded	default
User Interface	bmgr_pref_file	Specifies the location of the Graph Tool preference file.		
User Interface	button_name_in_help	Yes - The name and menu of any selected menu option displays, in English, in the help text associated with that button.	yes, no	no
User Interface	default_font	Sets font for text other than menu bar, menus and their children, and pop-up menus and help. Add comma-delimited variables in any order (italic bold, 24, times or 24, times, italic bold have the same effect). Any omitted variable uses standard setting.		
User Interface	dialog_translation	Specifies the language in which the dialog display when running non-English version of Creo Parametric.	yes, no	yes
User Interface	enable_popup_help	Controls pop-up help in dialog boxes. Yes - Help enabled. No - Help disabled.	yes, no	yes
User Interface	fonts_size	Determines the relative size of the text in menus and dialog boxes. You must restart Creo Parametric for the change to take effect.	medium, small, large	
User Interface	help_translation	Specifies the language in which the short help messages display when running non-English versions of Creo Parametric. Yes - Display in native language. No - Display in English.	yes, no	yes
User Interface	iconify_entire_pro	Determines how many other Creo Parametric windows/menus are iconified when one window is iconified. Yes - All the windows and menus are iconified. No - Only the selected window is iconified.	yes, no	yes
User Interface	kbd_cmd_abbreviation	On - Enables use of abbreviations when entering menu commands from the keyboard.	off, on	off
User Interface	kbd_selection	Provides the ability to select locations in the graphics area via the keyboard. Yes - Selection in the graphics area may be performed via the keyboard. No - Selection in the graphics area requires a mouse.	yes, no	no
User Interface	load_ui_customization_run_dir	Enables the loading of the UI customization file (.ui) if located in the run directory	yes, no	no
User Interface	mdl_tree_cfg_file	Specifies the Model Tree configuration file to be loaded when Creo Parametric is started.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
User Interface	menu_font	Specify font used in the Creo Parametric menu bar, menus, and all their children. Specify the comma-delimited variables in any order (italic bold, 24, times or 24, times, italic bold have the same effect). Any omitted variable uses the standard setting.		
User Interface	menu_manager_position	Controls the position of the Menu Manager. outside - Top left corner of Menu Manager is aligned to top right corner of window, inside - Top right corner of Menu Manager is aligned to top right corner of graphics area, adaptive - Menu Manager is positioned outside if there is enough room, otherwise it is positioned inside.	inside, outside, adaptive	adaptive
User Interface	menu_mapper_location	Specify your own URL for the Menu Mapper button in the help menu.		
User Interface	menu_translation	Specifies the language in which the menus display when running non-English version of Creo Parametric.	yes, no, both	yes
User Interface	modeltree_show_gen_of_instance	Determines whether the generic name is displayed as an extension of the instance name in the model tree and other model name lists.	yes, no	yes
User Interface	msg_translation	Specifies language in which Messages are displayed when running non-English version of Creo Parametric.	yes, no	yes
User Interface	native_kbd_macros	Specifies support of keyboard macros written in native language (for example, German).	yes, no	no
User Interface	new_parameter_ui	Enable the new parameter editor and user interface.	yes, no	yes
User Interface	new_relation_ui	Enable the new relations editor and user interface.	yes, no	yes
User Interface	online_resources_location	Specify your own URL for the Online Resources button in the help menu.		
User Interface	open_window_maximized	Open any new Creo Parametric window maximized by default. This config will override "reserve_menu_space" option.	yes, no	no
User Interface	param_dec_places	Sets the number of decimal places (0-14) to display in parameters.		6
User Interface	parameter_dialog_setup_file	Setup for the parameter dialog configuration file.		
User Interface	part_table_editor	Sets the editor that is used to edit notebook parameter tables. Protab - Use Pro/TABLE. Editor - Use the system editor (refer to pro_editor_command).	protab, editor, excel	
User Interface	pattern_relations_copy	Determines whether relations of pattern leader are inherited by pattern members.	yes, no	yes
User Interface	popuphelp_font	Specify font used in the Creo Parametric pop-up help. Add the comma-delimited variables in any order (italic bold, 24, times or 24, times, italic bold have the same effect). Any omitted variable uses the standard setting.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
User Interface	pro_editor_command	Enables use of an editor other than the system editor when the option editor has been specified as the value for the options part_table_editor or relation_file_editor.		
User Interface	prompt_on_exit	Determines if you are prompted to save objects when exiting a Creo Parametric session.	yes, no	no
User Interface	propagate_inactive_annotation	Yes - causes the propagation of inactive annotation.	yes, no	yes
User Interface	provide_pick_message_always	No - A message describing the (pre)selected item is provided only while querying. Yes - A message describing the selected item is provided in all cases (i.e. even when Query is not used).	yes, no	no
User Interface	relation_file_editor	Sets the editor used to edit relations.	protab, editor	editor
User Interface	reserved_menu_space	Reserves space for extra menus to the right of the Creo Parametric window (0.0-4.0). This option overrides windows_scale.		1
User Interface	right_mouse_button_popup	Specify whether or not the right mouse button pop up menu is available or not.	yes, no	yes
User Interface	selection_dragger_display_max	Threshold for the number of simultaneously selected items to not display draggers.		100
User Interface	set_menu_width	Determines width of menu manager panels. Default value is equivalent to 8 character widths. Acceptable range is from 8 to 20. Values lower than 8 and higher than 20 are ignored.		-1
User Interface	summary_help	Refers to the one-line help Messages that are either displayed or not displayed.	yes, no	no
User Interface	suppress_dlg_license_expired	Suppresses "License expired: List of options and expiration days" dialog		7
User Interface	suppress_license_loss_dialog	Yes - Suppresses the 'Regained Floating License' dialog box. No - The 'Regained Floating License' dialog box will display.	yes, no	no
User Interface	terminal_command	Specifies the full path to the terminal emulator command (the command that starts shell windows). Use the command for your system. Enter the complete path name and the terminal command.		
User Interface	thermo_position_hint	Enables you to position the thermometer-type scales whenever they appear, so that they do not overlap the Creo Parametric windows (if space permits, for example, if windows have been scaled).	no_window_overlap, window_overlap	window_overlap
User Interface	visible_mapkeys	No - Mapkeys will run without displaying picks being made in the Menu Manager menus. This does not affect the display of picks being made in menu bar menus or dialog boxes.	yes, no	no
User Interface	visible_message_lines	Sets the default number of visible message lines in the Creo Parametric message area.		2

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
User Interface	windows_scale	Scales Creo Parametric windows with a given coefficient. A value of 0.85 is usually adequate to allow dynamic menus to display to the right of the Creo Parametric window.		1
Weld	add_weld_mp	Yes - System includes welds when calculating mass properties and approximates lightweight weld mass properties. No - System excludes welds when calculating mass properties.	yes, no	no
Weld	pro_weld_params_dir	Specifies the directory to search when a weld parameter file is needed.		
Weld	weld_ask_xsec_refs	No - Do not ask for X-section references when creating weld features. Yes - Ask for X-section references when creating weld features.	yes, no	no
Weld	weld_color	Defines default weld color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using Tools>Colors>System.		0.000000 0.000000 0.000000
Weld	weld_dec_places	Sets the default number of decimal places (0-10) to display in weld parameters.		3
Weld	weld_edge_prep_driven_by	Determines if the edge preparation feature is created in the part or the assembly level.	part, assembly	part
Weld	weld_edge_prep_groove_angle	Specifies an initial default value for the angle cut edge preparation.		45
Weld	weld_edge_prep_groove_depth	Specifies an initial default value for the edge preparation depth.		0.25
Weld	weld_edge_prep_instance	Controls whether a family table instance is created for edge preparation. Yes - Instances of the components that receive edge preparation are created.	yes, no	
Weld	weld_edge_prep_name_suffix	Specifies the suffix name for the instance that will be created during edge preparation. Part name plus extension makes the instance name.		_noep
Weld	weld_edge_prep_root_open	Specifies an initial default value for the root opening edge preparation.		0.25
Weld	weld_edge_prep_visibility	Applies only if weld_edge_prep is set to Yes. Generic - Edge preparation features are resumed in the generic and suppressed in the instance. Instance - Edge preparation features are suppressed in the generic and resumed in the instance.	generic, instance	
Weld	weld_geom_type_default	Sets the default weld geometry type.	light, surface	surface
Weld	weld_notch_corner_radius	Specifies an initial default value for the radius at weld notch corner.		0.1

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Weld	weld_notch_height	Specifies an initial default value for the weld notch height.		0.4
Weld	weld_notch_radius	Specifies an initial default value for the weld notch radius.		0.5
Weld	weld_notch_width	Specifies an initial default value for the weld notch width.		0.5
Weld	weld_ui_standard	Specifies standard for welding user interface. ANSI - Use ANSI standards. ISO - Use ISO standards.	ansi, iso	ansi
Without category	accessory_window_above	Yes - Displays and keeps the accessory window above the active window. No - Does not keep the accessory window above the active window.	yes, no	no
Without category	accessory_window_display	Docked - Displays the accessory window embedded in the active window. Undocked - Displays the accessory window as a standalone window.	undocked, docked	docked
Without category	acis_out_version	Allows the user to choose ACIS export version.	4, 5	5
Without category	aec_parameter_file	Specifies the file name of the user defined piping parameter codes.		
Without category	allow_delete_many_in_drawings	Make the Delete Many command available in drawing mode. The Delete Many command is not supported by undo but can improve performance when deleting a significant number of items.	yes, no	no
Without category	allow_workpiece_silhouette	Determines if the workpiece is allowed for Silhouette Mach Window creation. NO - only reference part is allowed; YES - both workpiece and reference part are allowed; AUTO - only workpiece is allowed.	yes, no, auto	no
Without category	animation_imported_pbk_dir	Sets the default directory for the playback files to be used for Imported Animation.		
Without category	arc_radius_limit	Sets the upper limit for a radius in a CIRCLE statement of NCL file. The value must be positive. If not, 10*part_size is used.		-1
Without category	ask_designate_owners	Show the both required and optional owners to be designated.	yes, no	yes
Without category	assemble_module_in_rep	Retrieves the assembled module to CSA in a specified representation.		representative module variant
Without category	atb_search_path	Extends search path for ATB Update and ATB Check Status commands.		
Without category	attach_menumanager	Setup menu manager attachment to the main Creo Parametric window.	yes, no	no
Without category	auto_center_dimension	Automatically centers the dimension text between witness lines. For common reference dimensions, controls the centering of the first dimension only.	yes, no	no
Without category	auto_clean_shown_dimensions	This option controls whether the clean dimensions routine is automatically applied for newly shown dimensions. When set to NO the dimensions will be shown using their default locations from the model.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	auto_eval_current_cnfg_rule	If the option is "yes" simplified representation would be updated each time on retrieve/regeneration if it contains current configuration rule.	yes, no	yes
Without category	auto_show_3d_detail_items	When set to yes, AEs parallel to the view will be displayed while creating a new view.	yes, no	yes
Without category	auto_update_default_dwg_model	Automatically update the default drawing model to one shown in a view on the current sheet.	yes, no	yes
Without category	backup_reference_information	Yes-Backup the reference information of a feature once the feature regenerates successfully. No- Do not save the reference information of a feature.	yes, no	yes
Without category	bmx_param_restrictions	Restricted parameters automatically violate constraints in any Design Study	yes, no	yes
Without category	check_udf_int_key	Yes - updates UDF internal keys upon group's retrieval. No - skips keys update.	yes, no	yes
Without category	collect_dims_in_active_layer	This config options allows dimensions to be collected on the active layer.	yes, no	yes
Without category	combined_views_customization	Enable / Disable combined views customization.	yes, no	yes
Without category	comp_assemble_with_interface	Settings for assembling component with interface(s).	none, default_multi, default_single, from_list	default_multi
Without category	comp_interface_placement	Sets the default interface placement type. Interface to geometry - place the component using component interfaces and assembly geometry. Interface to Interface - place the component using assembly and component interfaces.	interface_to_interface, interface_to_geom	interface_to_interface
Without category	comp_normal_offset_eps	Specifies the angle epsilon such that if the desired surfaces are equal or less than the epsilon (in degrees), then a normal constraint will be created.		10
Without category	comp_placement_legacy_move	Show the legacy Move tab in the Component Placement dashboard	yes, no	yes
Without category	comp_snap_angle_tolerance	Specify the angle tolerance for snapping while freeform dragging components. Default value is 30.0 degrees.		30
Without category	comp_snap_dist_tolerance	Specify the distance tolerance for snapping while freeform dragging components. Default value is 0.1 (relative to size of component being assembled).		0.1
Without category	confirm_on_edit_foreign_models	Provides a confirmation dialog when attempting to make design changes to foreign models	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	constraint_reference_redefine	keep_constraint_type - Adjusts the component position to keep the same constraint type. reset_constraint_type - Automatically changes the constraint type to keep the position of the component.	keep_constraint_type, reset_constraint_type	keep_constraint_type
Without category	contact_surfaces_gap	Sets the maximum allowable gap between weld contact surfaces.		0
Without category	convert_to_wf3_replace	Convert to Wildfire3 replace functionality.	yes, no	no
Without category	cosmetic_thread_match_tolerance	Sets the tolerance, which is used in the analysis to match standard thread type and standard radius, to the selected thread surface.		0.001
Without category	create_drw_sheet_thumbnails	Controls whether thumbnail images of drawing sheets will be generated and displayed with the drawing sheet tabs. Setting this option to No may improve performance with large drawings.	yes, no	yes
Without category	creotkdat	Specify the full path to an additional Toolkit registry file.		
Without category	default_cable_axis_location	Sets the location type for axis. Choose Along to create one location at each end of the axis.	on, along	on
Without category	default_geom_analysis_type	Default save type for geometry analysis.	quick, saved, feature	quick
Without category	default_ref_current_module_vrnt	Allow references to current design solution only.	yes, no	yes
Without category	des_exp_folder	Sets the directory for the Design Exploration session. The default location is in a default system folder. Type a valid file path to change the directory to a user defined one.		
Without category	designate_model_item_params	When an object is designated, all parameters belonging to that object will be automatically designated as well.	yes, no	no
Without category	disp_in_open_rep_create	Simplified Rep to display initially in Open Rep Create: "No" for none, "Yes" for graphics rep, or "Default Rep" to display the rep indicated by option "open_simplified_rep_by_default".	yes, no, default rep	no
Without category	display_all_dims	Controls the display of dimensions during dynamic edit.	yes, no	yes
Without category	display_combined_views	Display combined view tabs by default	yes, no	no
Without category	display_images	Display images outside of Style feature	yes, no	yes
Without category	display_mapkeys_dialog_on_top	Controls whether mapkeys dialog will be displayed always on top (yes) or behind the main window (no).	yes, no	no
Without category	display_pdm_server_quick_view	Yes - display summary view of PDM information for items in session. It applies to ProductPoint only.	yes, no	yes
Without category	display_popup_viewer	No- disable popup viewer, File_Open_Only: display popup viewer only in the file open dialog, Yes: display popup viewer in file open dialog, model tree and in Server Quick View dialog	yes, no, file_open_only	file_open_only
Without category	display_sketch_dims	Controls the display of sketch dimensions during dynamic edit.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	dm_enable_simple_import_export	Enable a simplified method of import/export between disk and Workspace.	yes, no	yes
Without category	dm_nc_process	Sets output of NC process information as content on the NC Assembly CAD Document in XML Format. Yes - Outputs the file whenever CAD Document is Saved. No - Does not output the file.	yes, no	no
Without category	dm_nc_references	Sets whether reference relationships will be created in Windchill between the manufacturing model and its deliverables(CL file, tape file and DMIS file).	yes, no	yes
Without category	dm_preview_searches_server	Search for files on the server while previewing.	yes, no	yes
Without category	drw_prompt_for_combined_state	Determines whether the combined state prompt will appear in a drawing when creating a view.	yes, no	yes
Without category	dwg_unicode_conv_lang_scope	Specifies scope of the option 'dwg_unicode_conversion_language'. '2D' to apply to 2D models only, 'ALL' to apply to all models.	all, 2d	2d
Without category	edge_tess_quality	Set the accuracy level for arc tessellation.	high, medium, low	low
Without category	enable_3dmodelspace_browser_tab	Yes - Display 3D ModelSpace tab in the browser when Creo Parametric starts. No - Display 3D ModelSpace tab only when you click a 3D ModelSpace link in the navigator.	yes, no	yes
Without category	enable_face_preselection	Controls the ability to "pre-select" faces/datums as an entry to Sketch based features such as Extrude, Revolve (input to the internal sketch).	yes, no	yes
Without category	enable_face_thread_turning	Allows/disallows face option for thread turning sequence.	yes, no	no
Without category	enable_fsaa	Enable OpenGL full screen anti-aliasing.	off, 2, 4, 8, 16, 32	off
Without category	enable_obsoleted_features		yes, no	no
Without category	enable_offset_fan_curve	Enables fan curve option for offset tool.	yes, no	no
Without category	enable_opengl_shader	Enable OpenGL Phong lighting.	yes, no	yes
Without category	esr_mass_properties	Defines whether to include all immediate components of external simplified rep into Mass Properties calculations or only defined state of design assembly.	design_only, extended	design_only
Without category	expand_all_with_nested_layers	Yes - expand all including nested sub layers.	yes, no	yes
Without category	fasthlr_drawing	Specifies whether fast hidden line removal is on/off by default.	yes, no	no
Without category	file_dialog_units_class	Sets the units for manikin attributes displayed in the File Open dialog.	mks, ips	mks

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	file_timestamp_format	Controls the format of time stamps in file dialogs and the folder browser. The format for the setting is a string consisting of up to seven portions: the year, the month, and the date, the hours, minutes, seconds and am/pm designator. You can enter the portions in any order.		%dd-%mmm-%yy %hh:%mi:%ss %pm
Without category	flex_display_dflt_bounding_e dge	Turns on/off highlighted display of default bounding edges in flexible modeling features that can have multiple solutions.	yes, no	yes
Without category	flex_dma_reduce_solution_n umber	Reduces the number of default solutions found in Flex features, when modifying geometry with mixed convexity contours.	yes, no	yes
Without category	flex_tan_propagation_default	Defines default state of the Tangency Propagation option in Flexible Modeling features.	off, on	off
Without category	group_repl_with_recycle	Yes - enables new UDF replacement with family table instances that prevents children failures. No - UDF replacement will work in the same manner as Wildfire 3.0 and earlier releases.	yes, no	yes
Without category	group_replace_rename	Yes - updates replaced group name with replacement group name. No - preserves existing name of the group being replaced if it is a user-given name.	yes, no	yes
Without category	hdic_export_v5_secondary_c gr	In HDIC export for CATIA V5 allows to attach CGR as a secondary content	yes, no	no
Without category	hdic_export_v5_to_ws_enabl e	Enables HDIC export for CATIA V5 to Workspace.	yes, no	no
Without category	idd_repair_tangency	Sets the default state of the "Repair Tangency" checkbox in the Repair and Match tools	yes, no	no
Without category	include_all_tools_in_list	Show/hide inapplicable cutting tools in mfg sequence UI.	yes, no	no
Without category	intf_nx_version	Sets the version of NX interface.	6, 7, 8	6
Without category	intf3d_out_catia_v5_version	Allows the user to switch between the default and alternate converter version for export to CATIA V5	16, 17, 18, 19, 20, 21, 22	16
Without category	intf3d_out_nx_version	Allows the user to switch between the default and alternate converter version for export to NX	3, 5, 7	3
Without category	jlink_java_command	Specifies a command to start JRE for J-Link. May contain arguments and environment variables. Overrides the default, see also jlink_java2.		
Without category	js_error_policy	Control the display and execution of the java script error dialogs. If the dialog is presented, the user can choose whether to stop or continue running the script.	show, suppress_continue, supress_stop	
Without category	keep_model_in_open_windo w	Controls whether a model is displayed in its window when the accessory window is open.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	last_session_directory_path	Sets the directory that holds the temp files for the session recording mechanism.		<work_dir>\appdata\roaming\ptc\proengineer\wildfire\wf\settings
Without category	last_session_retrieval	Yes - Activates recording mechanism that enables the session to be retrieved during the next session. No - The recording mechanism is not available and the session can not be retrieved.	yes, no	yes
Without category	layer_rule_retrieve_objects	This option controls whether or not the system will retrieve additional objects needed to evaluate a layer rule. OK tells the system to retrieve the objects. Cancel stops evaluation of the rule.	ok, cancel, prompt	prompt
Without category	layout_data_sharing	Allows to change the default ce entity type to be public or private.	private, public	private
Without category	layout_guides	Enable/Disable layout guides.	yes, no	yes
Without category	layout_precision_decimals	Display 0 to 5 decimal places in precision panels	1, 3, 2, 4, 5	2
Without category	layout_precision_panel	Yes - Precision Panel will be shown; No - Precision Panel will not be shown.	yes, no	yes
Without category	layout_smart_constraints	To improve the user experience, specific entity types are appropriately constrained by default (i.e. - fillets with tangency constraints). Yes - specific entity types will be created with appropriate constraints by default; No - constraints will never be automatically created for any entity type.	yes, no	yes
Without category	layout_symbol_inst_palette_file	Specifies the location of the layout symbol instance palette. If you do not specify a directory, layout will use the same palette as Creo Parametric drawings. Use full path name to avoid problems.		
Without category	layout_template_dir	Specifies the default directory for layout template files. Use full pathname to avoid problems.		
Without category	lda_batch_mapping_option	Specifies the options for mapping in LDA from dBatch.	all, model_only, views_only, annotations_only, model_and_views, views_and_annotations	all
Without category	lda_map_file	Path to the file containing the list of matching drawings and solid models.		
Without category	lda_model_dir	Path to the directory containing the solid models.		
Without category	lda_overall_view_map_time_limit	Specifies LDA overall view mapping time limit in seconds.		300
Without category	lda_prefix_in_basic_dim_box	Add prefix to basic box while converting the dimension to basic.	yes, no	no
Without category	lda_show_entities_default	Specify whether to show mapped draft entities by default in Validate Association Dialog.	yes, no	no

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	lda_view_map_time_limit	Specifies LDA single view mapping time limit in seconds.		60
Without category	manikin_drag_auto_locks	Manikin drag auto locks	yes, no	yes
Without category	manikin_drag_legacy_rotate	Controls the Manikin Rotate Around Axis drag mode: Legacy (YES) or direct joint angle control (NO)	yes, no	no
Without category	manikin_flexibility	Sets the level of flexibility that is allowed when dragging a manikin. FULL - All Manikin segments are articulated. NONE - Manikin is calculated as a single body. DEFAULT - Manikin finger segments are locked to a single body.	none, default, full	default
Without category	manikin_replace_keep_posture	Controls whether a manikin's existing posture is applied to a replacement manikin.	yes, no	yes
Without category	mass_prop_load_from_file	Controls whether to take into the account values from mass properties file.	by_request, always	by_request
Without category	measure_auto_replace_mode	Automatically switches to Replace mode for the last reference in appropriate Measures (Distance, Angle, Transform)	yes, no	no
Without category	measure_dec_places	This configuration option controls the number of digits after the decimal point. If value = -1, this configuration is ignored and measure_sig_figures is used		-1
Without category	measure_dialog_expand	Determines default look of measure dialog (expanded or collapsed)	yes, no	no
Without category	measure_ref_option_status	Determine As Object (As Center et al.) checkbox option behavior in Measure Tool	default, all_on, all_off, keep_last	default
Without category	measure_show_featab	Controls initial display of the Feature tab in Measure Tool. Yes - feature tab is always displayed by default. No - feature tab state is saved for the duration of the session upon closing of the dialog.	yes, no	yes
Without category	mfg_cl_euc_japanese_encoding	Yes - Output cl-file Japanese characters in EUC encoding.	yes, no	no
Without category	mfg_paste_special_dep_default	Controls the default state of the following option in Paste Special dialog: "Make copies dependent on dimensions of originals". This configuration option has an effect only for manufacturing assemblies.	yes, no	yes
Without category	mfg_tap_mat_rem_coef	The factor is used to consider the threading of tapping tool. The estimate removed material value of tapping sequence is presented as estimate removed material value multiplied by this factor. The factor should be greater than 0 and upto 1.0.		1
Without category	model_tree_font	Specify font used in the Creo Parametric Model Tree. specify the comma-delimited variables in any order (italic bold, 24, times or 24, times, italic bold have the same effect). Any omitted variable uses the standard setting.		

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	modelcheck_dir	Specifies path to the ModelCHECK configuration files.		
Without category	module_variant_cross_refs	Allow references between design solutions in Module	yes, no	no
Without category	mp_analysis_default_source	Specifies the default source for mass properties calculation via Analysis.	geometry, assigned, mp_source	
Without category	mp_calc_level	Specifies the level of mass properties calculation.	assembly_only, all_models	assembly_only
Without category	multiple_skeletons_allowed	Allows/restricts multiple skeleton.	yes, no	no
Without category	nc_jobman_visible	Yes - make "Submit Path" and "Abort Path" visible. No - make "Submit Path" and "Abort Path" invisible.	yes, no	no
Without category	new_ref_part_name_disabled	Disallow redirecting data sharing feature missing reference model to a model with different name.	yes, no	yes
Without category	old_style_set_datum	Yes - datum properties dialog has button for old-style set datum. No - datum properties dialog does not have button for old-style set datum.	yes, no	yes
Without category	open_builder_in_mt_mode	Open the variant builder in Model Tree Selection mode	yes, no	no
Without category	orientation_style	Set spin orientation style to dynamic or anchored.	anchored, dynamic	dynamic
Without category	output_mfg_xml	Allows export/import mfg model to/from xml file. Yes - make Export/Import options visible. No - make them invisible.	yes, no	no
Without category	pdm_server_quickview_rows_num	Sets the default number of rows for the pdm quick view list.		7
Without category	plot_names_with_sheet	Yes - Plot file names are appended with _(sheet number) before file extension.	yes, no	no
Without category	popup_refpairintch_window	Bring forward the window being used with the reference pairing table.	yes, no	no
Without category	prime_analysis_directory	Sets the default directory for the Prime files to be used for Prime analysis calculations. Use the full pathname to avoid problems.		
Without category	prime_automapping	Toggles Prime analysis automapping of variables	yes, no	no
Without category	propagate_by_default	Yes - If the modified geometry belongs to a geometry pattern, a pattern recognition feature or a symmetry recognition feature, the modification is automatically propagated to all pattern/symmetry members.	yes, no	no
Without category	ptc_manikin_comforts_path	Sets the default directory for the Creo Parametric manikin comfort angles. You must restart Creo Parametric for the change to take effect.		
Without category	ptc_manikin_library_path	Sets the default directory for the Creo Parametric manikin library (or custom manikin libraries you built containing the proper index menu files). Use the full path name to avoid problems. You must restart Creo Parametric for the change to take effect.		<creo3.0_loadpoint>\manikin_lite\

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	ptc_manikin_posture_path	Sets the default directory for the Creo Parametric manikin postures (or custom posture libraries you built). Use the full path name to avoid problems. You must restart Creo Parametric for the change to take effect.		
Without category	qty_balloon_change_warning	Sets the warning style when quantity balloons have changed in a drawing. Dialog displays the warning in a dialog box, Message displays the warning in the message area only.	dialog, message	dialog
Without category	quick_print_drawing_template	Specify the path and name of a drawing template to be used by the File > Quick Print command.		
Without category	quick_print_plotter_config_file	Specify the path and name of the default plotter config file to be used by the File > Quick Drawing command.		
Without category	radial_hole_linear_dim	YES - Option for creation of radial hole with a linear dimension will be available.	yes, no	no
Without category	record_mapkey_comment_to_trail	Controls whether a comment is written to the trail file whenever a mapkey is executed.	yes, no	yes
Without category	regen_failure_handling	Specifies whether to enter resolve mode when regeneration failures occur. Resolve_mode - Enter resolve mode when regeneration failures occur. No_resolve_mode - Don't enter resolve mode when regeneration failures occur.	resolve_mode, no_resolve_mode	no_resolve_mode
Without category	regen_int_mdls_on_retr	Yes to regen models intersected with assem feats during assembly retrieval.	yes, no	no
Without category	regen_solid_before_save	Controls whether to regenerate top model, or prompt user before regeneration if it is required	yes, no, prompt	prompt
Without category	relations_ang_eval_diff_check	Check for differences in the evaluation of relation sets evaluated and saved in versions of Creo 2.0 prior to M110. Warn_message Check and give a warning in the message area. Warn_dialog Check and give a warning in a message window. Ignore Skip the check and warning.	ignore, warn_message, warn_dialog	warn_message
Without category	relations_num_const_units	Specifies whether diagnostic should be issued when an unitless numeric value causes relation expression evaluation trouble.	yes, no	no
Without category	relations_units_sensitive	Set the default unit sensitivity (ignore or take into account parameter/dimensions units) in creation of new relation data (legacy relation data will not be unit sensitive until "converted" to one).	yes, no	yes
Without category	remember_last_attach_type	Yes - Sets the default of the ATTACH TYPE menu of the last options picked.	yes, no	yes
Without category	remember_replaced_components	Set to "Yes" this option will preserve a dependency to the replaced, outgoing, component. Set to "No" the dependency will, when possible, be removed.	yes, no	yes

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	replace_comp_notes_method	Specify treatment of notes when components are replaced. Transfer - transfer to new component. Remove - remove the notes. Preserve - save notes in case component is restored.	transfer, remove, preserve	transfer
Without category	reroute_affected_children	Affected children when setting Current solution.	none, auto, selective	selective
Without category	restricted_val_definition	Specify name and path of the file contained restricted values definition.		
Without category	retrieve_display	Controls whether or not graphics data is loaded in the background.	background, foreground	background
Without category	retrieve_merge_ref_parts	Automatic reference parts retrieval for dependent Merge.	yes, no, ignore_missing	no
Without category	rmv_xsec_unlinked_cosm_features	Specify whether to automatically try to delete, or to detach from the feature list, the internal sketch features which were unlinked from offset cross sections, and whether to do it only upon sketch unlinking, or upon model retrieval too.	none, detach_on_unlink, delete_on_unlink, detach_on_both, delete_on_both	detach_on_unlink
Without category	round_prewf5_displayed_dimension_val	Rounds the displayed value	round all, round none, calculated	calculated
Without category	save_view_states_method	Setup the method used on saving view states changes.	prompt, save changes, discard changes	prompt
Without category	shade_with_edge_tangent	Display Tangent edges in Shade with Edge view mode.	yes, no	yes
Without category	show_affected_children_dialog	Do not invoke dialog for setting Current solution.	yes, no	yes
Without category	show_axes_by_view_scope	Determines the scope of the Show Model Annotations tool when showing model datums and selecting a drawing view. For large assembly drawings, use top_model_only to improve performance by restricting the set of axes to be shown.	top_model_only, all_sub_models	top_model_only
Without category	show_description_view_mgr	Yes - show a column for Description field for all tabs in view manager.	yes, no	no
Without category	show_object_type_in_taskbar	Yes - Displays the icon of the open PTC Creo object in the taskbar and window title bar. For all PTC Creo applications except PTC Creo Parametric, icons include a letter indicating the application name. No - Displays the PTC Creo application icon in the taskbar and window title bar.	yes, no	no
Without category	show_sketch_dims_in_feature	Sets the default display state of internal sketch dimensions in the feature definition environment.	yes, no	no
Without category	simplprep_column_display	Controls display of Simplified Representations columns in Model Tree, 'Automatic'-'Current Rep' column is shown automatically when a rep is activated, 'Manual' visibility of the column is fully controlled by user, pre-Wildfire 6 behavior.	automatic, manual	automatic

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	simprep_column_format	Specifies simplified representation model tree column format.	names only, icons only, icons and names	icons and names
Without category	simprep_default_cnfg_asm	Retrieves a CSA model in a specified representation. Yes - uses Open Rep dialog. No - retrieves the Master representation. Specified Name - retrieves the model in the specified representation name if it exists or the Default representation.		last stored
Without category	simprep_ondemand_activation	Enables retrieving on-demand the Master rep of an activated model. When an assembly is activated the Assembly only rep is retrieved.	yes, no	yes
Without category	simprep_ondemand_cleanup	Determines if a model retrieved on demand should be removed or erased. Disabled - the model will not be removed and will remain in the assembly. Removed - the model will be removed. Remove_Erase - The model will be removed and erased from memory.	disable, restore, restore_and_erase	restore
Without category	simprep_ondemand_editing	Enables on demand retrieval of an edited item and its parent references. Configuration tools (Family Tables, Relations, etc) will retrieve required models accordingly. Automatic - Retrieves the minimum rep required to perform the operation.	automatic, master, disable, master_with_ancestors	automatic
Without category	simprep_ondemand_regeneration	Enables on demand retrieval of required models during regeneration. Can affect regeneration performance.	automatic, master, disable	automatic
Without category	single_side_surf_color	Allow application of color to a single side of a surface.	yes, no	no
Without category	sketcher_curve_color_synch	Synchronize color assignment between sketched curves and their sketches.	yes, no	yes
Without category	sketcher_palette_path	Sets the default directory for the user sketcher shape library. Use the full path name to avoid problems.		
Without category	sketcher_rel_accuracy	Enter a sketcher relative accuracy.		1
Without category	sketcher_strngthn_to_def_def_c_pl	This option enable the user to keep a highly precise value during creation or conversion strong dimension.	yes, no	yes
Without category	snap_single_constraints	When dynamically dragging components during component placement, allow to snap to references one constraint at a time.	yes, no	yes
Without category	spin_with_orientation_center	Display orientation center while reorienting.	yes, no	yes
Without category	style_state_transparency	Set the value of transparency for components set to "Transparent" in style state. Value must be between 0 and 100.		70
Without category	suppress_dim_ghost_with_ctrl	Suppress the ghost image of dimension when CTRL is pressed. It allows better visibility while selecting further dimension references.	yes, no	no
Without category	template_cnfg_asm	Specifies the default configurable assembly template file. Use full pathname to avoid problems.		inlbs_csa.asm

## PTC Creo 3.0 Parametric Configuration Options

Category	Name	Description	Values	Default Value
Without category	template_esrasm	Specifies the designated template ESR assembly. Use full path to avoid problems.		inlbs_asm_esr.asm
Without category	template_layout	Specifies the default layout template file. Use full pathname to avoid problems.		
Without category	tk_enable_ribbon_custom_save	Enable storing the Auxiliary Application User Interface customization	yes, no	no
Without category	tool_mgr_save_pocket_in_xml	Determines whether the tool position should be saved in a XML file (*.txf) created in tool manager dialog.	yes, no	yes
Without category	travel_bounds_display	Enable travel bounds display.	yes, no	no
Without category	travel_bounds_transparency	Transparency value for machine bound display.		0.8
Without category	trngrv_roundtool_nobackcut	Yes - Groove turning using round tool generates no back cut motion for profile pass.	yes, no	no
Without category	upd_param_on_retr_mark_modified	Allows/disallows mark model as changed if as a result of mass properties/reasons calculation during retrieval some parameter values are changed.	yes, no	no
Without category	use_master_for_bad_disp_reps	Yes to retrieve master rep when graphics rep has no display data.	yes, no	no
Without category	use_strict_ui_trail	Perform extra validation of user interface actions when playing back trail files.	yes, no	yes
Without category	use_temp_interface_for_paste	Yes - Enables use of interfaces for multiple placement of components during paste.	yes, no	yes
Without category	use_web_page_font_in_reports	Controls whether HTML reports use the browser's standard fonts.	yes, no	no
Without category	visible_recent_files	Number of visible recent files. The possible range is 1 - 30.		21
Without category	web_enable_javascript	Enables or disables Pro/Web.Link in the embedded browser.	off, on	off
Without category	web_max_mail_size	Set the maximum email attachment size in kilobytes.		5000
Without category	weld_fillet_preferences_file	Sets the path and name of the fillet weld preferences file.		
Without category	weld_general_preferences_file	Sets the path and name of the general weld preferences file.		
Without category	weld_groove_preferences_file	Sets the path and name of the groove weld preferences file.		
Without category	weld_plug_slot_preferences_file	Sets the path and name of the plug/slot weld preferences file.		
Without category	weld_spot_preferences_file	Sets the path and name of the spot weld preferences file.		

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Assembly	allow_freeze_failed_assy_comp	NEW	Allow freezing failed assembly components.	yes, no	no
Assembly	auto_backup_new_placemnt_refs	DELETED	Yes - Create Component Reference Backup, No - Do not Create Component Reference Backup.	yes, no	no
Assembly	freeze_failed_assy_comp	DELETED	No - Requires an action to fix the assembly or freeze the component that fails retrieval. Yes - Freezes any component failing retrieval at its last known assembly location.	yes, no	no
Creo Simulate	sim_hyperelastic_material_fit	CHANGED	Sets the default method for calculating coefficients for hyperelastic materials. If set to "normalized", normalized stresses will be used. If set to "non-normalized", non-normalized stresses will be used.) ----- Sets the default method for calculating coefficients for hyperelastic materials. If set to "normalized", normalized stresses will be used. If set to "non-normalized", non-normalized stresses will be used.	normalized, non-normalized	normalized
Data Exchange	afx_enabled	CHANGED	Loads EFX application. ----- Loads AFX application.	yes, no	no ----- yes
Data Exchange	atb_show_foreign_name	NEW	Displays foreign source model name in the model tree for imported models.	yes, no	yes
Data Exchange	board_bend_notes_dflt_display	NEW	Define the default state of bend note display (board mode)	yes, no	yes
Data Exchange	board_bend_notes_direction_down	NEW	Defines the down direction symbol for board bend notes.		default
Data Exchange	board_bend_notes_direction_up	NEW	Defines the up direction symbol for board bend notes.		
Data Exchange	board_bend_notes_order	NEW	Defines the fields order for board bend notes.		&type&direction&angle
Data Exchange	board_bend_notes_type_formed	NEW	Defines the formed symbol for board bend notes.		default
Data Exchange	board_bend_notes_type_rolled	NEW	Defines the rolled symbol for board bend notes.		default
Data Exchange	board_bend_table_directory	NEW	Will be used to set the bend allowance table only. Sets the default directory of the board bend allowance table files. If not set, the system supplied bend tables are used. Use the full name and path to avoid problems.		

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Data Exchange	board_design_rules_directory	NEW	Will be used to set the design rules file folder only. Will be used by the design rules manager. Sets the default directory of the board design rules files. Use the full name and path to avoid problems.		
Data Exchange	board_drive_bend_by_parameters	NEW	Board bend radius and bend angle values and bend dimension location will be driven by feature-level relations to board parameters.	yes, no	no
Data Exchange	board_drive_tools_by_parameters	NEW	Drives the board options and values by feature-level relations to board parameters.	yes, no	yes
Data Exchange	board_flange_shape_sketches_dir	NEW	Sets path to directory that contains the Flange shapes.		
Data Exchange	board_flat_shape_sketches_dir	NEW	Sets path to directory that contains the Flat shapes.		
Data Exchange	board_parameters_directory	NEW	Specifies the directory to save/retrieve board parameters files. Use full name and path to avoid problems.		
Data Exchange	dwg_export_format	CHANGED	Provides the ability to choose DWG file version when exporting from Creo Parametric drawing.	14, 2000, 2004, 2007, 2010 ----- 14, 2000, 2004, 2007, 2010, 2013	2007 ----- 2013
Data Exchange	dxf_export_format	CHANGED	Provides the ability to choose DXF file version when exporting from Creo Parametric drawing.	12, 13, 14, 2000, 2004, 2007, 2010 ----- 12, 13, 14, 2000, 2004, 2007, 2010, 2013	2007 ----- 2013
Data Exchange	dxf_in_proxy_entity	DELETED	Controls the input support for ACAD_PROXY_ENTITY.	yes, no	yes
Data Exchange	ecad_edmd_out_version	CHANGED	Set the EDMD output version.	2.0, 1.2 ----- 2.0, 1.2, 3.0	2
Data Exchange	ecad_outline_ents_enhanced_exp	CHANGED	Enhanced export of special entities to ECAD formats. Yes - Attempt to approximate arcs and lines. No - Use part accuracy to approximate lines.	yes, no	no ----- yes
Data Exchange	ecadcollab_accept_color	NEW	ECAD Collaboration Accept color		0.0 255.0 0.0
Data Exchange	ecadcollab_auto_redef_areas	NEW	Automatically redefine ECAD Areas affected by collaboration changes	yes, no	yes

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Data Exchange	ecadcollab_auto_redef_comps	NEW	Automatically redefine components affected by collaboration changes	yes, no	yes
Data Exchange	ecadcollab_auto_redef_holes	NEW	Automatically redefine holes affected by collaboration changes	yes, no	yes
Data Exchange	ecadcollab_preview_after_color	NEW	ECAD Collaboration Preview After color		255.0 127.0 0.0
Data Exchange	ecadcollab_preview_before_color	NEW	ECAD Collaboration Preview Before color		0.0 76.0 255.0
Data Exchange	ecadcollab_preview_level	NEW	ECAD Collaboration preview Level	full, partial, light	partial
Data Exchange	ecadcollab_reject_color	NEW	ECAD Collaboration Reject color		255.0 127.0 127.0
Data Exchange	ecadcollab_scan_des_on_area_chg	NEW	Check for affected objects resulting from ECAD Areas collaboration changes	yes, no	yes
Data Exchange	ecadcollab_scan_des_on_board_chg	NEW	Check for affected objects resulting from board collaboration changes	yes, no	yes
Data Exchange	ecadcollab_scan_des_on_component_chg	NEW	Check for affected objects resulting from components collaboration changes	yes, no	yes
Data Exchange	ecadcollab_scan_des_on_hole_chg	NEW	Check for affected objects resulting from holes collaboration changes	yes, no	yes
Data Exchange	ecadcollab_supp_failed_features	NEW	Suppress of failed features on Object changes	yes, no	yes
Data Exchange	ecadcollab_suppress_cuts	NEW	Automatically suppress cut features in case of board collaboration changes	yes, no	yes
Data Exchange	initial_board_bend_y_factor	NEW	Specifies a value used to determine the neutral bend line for a board part. This value is always used for non-cylindrical bends. It is used for regular bends only when no bend table is specified		0.5
Data Exchange	intf_ug_version	CHANGED	Sets the version of Unigraphics interface. ----- Sets the version of NX interface.	nx6, nx7, nx8	nx6
Data Exchange	intf2d_in_acad_expl_unsupported_entities	NEW	Yes - ACAD_PROXY_ENTITY and REGION entities are imported as set of draft entities, No - ACAD_PROXY_ENTITY and REGION entities are ignored.	yes, no	yes
Data Exchange	intf2d_in_mi_brk_unsupported_dimension	NEW	Yes - Breaks unsupported MI dimension types to separate entities, No - Imports unsupported MI dimension types as different type dimensions that can look differently.	yes, no	no



## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Drawing	drawing_setup_file	CHANGED	Sets the default drawing setup file option values for your Creo Parametric session, otherwise, the system uses the default drawing setup file option values. Certain parameters in the file are valid only if you have a license for Pro/DETAIL. -----		<creo2.0_loadpoint>\text\prodetail.dtl ----- <creo3.0_loadpoint>\text\prodetail.dtl
Drawing	pro_palette_dir	CHANGED	Sets the default directory for drawing symbol palette files.		<creo2.0_loadpoint>\symbols\palette \ ----- <creo3.0_loadpoint>\symbols\palette \
Drawing	snap_drw_entity_to_snap_line	CHANGED	Yes - Enable snapping of drawing and report entities to snap lines. No - Disable snapping of drawing and report entities to snap lines. ----- Yes - Enable snapping of drawing and report entities to snap lines. No - disable snapping of drawing and report entities to snap lines.	yes, no	yes
Education & Social Tools	enable_sociallink	DELETED	Allow Windchill SocialLink to connect with Creo Parametric. When enabled SocialLink commands will be added to the Application Menu and the Ribbon bar. Enter "Yes" to enable SocialLink, and "No" to disable it	yes, no	no
Electromechanical	bundle_transition_create_method	NEW	Control whether bundle transitions are created automatically. Default is automatic.	automatic, manual	
Electromechanical	cable_bundle_transition_length	NEW	Transition length is controlled by the formula Larger Diameter X <percentile>. Default value is 50%#91;0...250]%. -----		0.5
Electromechanical	update_loom_bundle_data	NEW	Setup whether to update loom bundle data from old harnesses.	yes, no	no
Environment	activate_window_automatically	NEW	Yes - Execute Activate command automatically after a window is selected. No - Don't execute Activate command automatically after a window is selected.	yes, no	yes
Environment	des_exp_confirm_on_save	NEW	When you use a `Save` command during a Design Exploration session, it does not save the session, it prompts to add a new checkpoint.	yes, no	yes

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Environment	des_exp_create_auto_cps	NEW	Set the maximum number of automatically created checkpoints allowed. Additional checkpoints override previous checkpoints. Type a value from 1 to 99. Type 0 to prevent automatic checkpoint creation.		
Environment	des_exp_create_backups	NEW	Back up all models in session when starting Design Exploration and all models opened and modified during the session. When backup is on, starting a Design Exploration session might take some time.	yes, no	no
Environment	des_exp_data_path	NEW	Sets the directory for the Design Exploration session. The default location is in a default system folder. Type a valid file path to change the directory to a user defined one. Data from only one session is stored.		
Environment	des_exp_display_type	NEW	Set the default display of the Checkpoint Tree. 'Tree' - Displays all branches in the Checkpoint Tree. 'Timeline' - Displays checkpoints in chronological order ( <u>des_exp_display_type</u> ).	tree, timeline	tree
Environment	des_exp_show_cp_props_dialog	NEW	Open the Checkpoint Properties dialog box when you add a new checkpoint, or turn an automatic checkpoint into a regular one.	yes, no	yes
Environment	intelligent_fastener_enabled	NEW	Enables Intelligent Fastener. You must restart Creo for the setting to take effect.	yes, no	yes
Environment	prehighlight_tree	CHANGED	Default value for "Preselection Highlighting" option in the model tree, layer tree and detail tree. When enabled, selectable items are highlighted prior to being selected in the tree. ----- Default value for "Preselection Highlighting" option in the model tree, layer tree, detail tree and layout tree. When enabled, selectable items are highlighted prior to being selected in the tree.	yes, no	no
Features	auto_add_remove	CHANGED	Allows Creo Parametric to automatically switch from add to remove of material for solid extrude and revolve during feature creation.	yes, no	no ----- yes
Features	flex_maintain_solution_topology	NEW	Initial value for the "Maintain solution topology" option in created flexible modeling features.	yes, no	no

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
File Storage & Retrieval	pro_font_dir	CHANGED	Sets the default font directory (Creo Parametric will load all fonts in this directory into the drawing regardless of the contents of the system and working directories).		<creo2.0_loadpoint>\text\fonts\ ----- <creo3.0_loadpoint>\text\fonts\ -----
File Storage & Retrieval	pro_surface_finish_dir	CHANGED	Sets the default directory for your user-defined surface finish symbols. Use the full pathname to avoid problems.		<creo2.0_loadpoint>\symbols\surffin s\ ----- <creo3.0_loadpoint>\symbols\surffin s\ -----
File Storage & Retrieval	save_dialog_for_existing_models	CHANGED	Controls whether the Save dialog is shown for models whose storage location is already known.	yes, no	yes ----- no
File Storage & Retrieval	save_file_iterations	NEW	Yes - Iterations of a file will be saved. No - Iterations of a file will not be saved.	yes, no	yes
File Storage & Retrieval	template_boardpart	NEW	Specify the model to use as the default new ECAD board template.		
File Storage & Retrieval	template_new_ecadasm	NEW	Specify the model to use as the default new ECAD assembly template.		
File Storage & Retrieval	retrieve_data_sharing_ref_parts	CHANGED	Automatic reference parts retrieval for dependent data sharing features.	yes, no, ignore_missing	no
Update Control					
Freestyle Feature	freestyle_drag_performance	NEW	Yes - Improve performance during drag by temporarily reducing graphics quality. No - No reduction in graphics quality during drag.	yes, no	yes
Layout	layout_feat_copied_geom_thickness	NEW	Layout feature - Copied geometry thickness.	1, 3, 2, 4, 5, 1.5, 2.5, 3.5, 4.5	1.5
Manufacturing	enable_param_n_tool_upd_dialog	CHANGED	Enables UI for modification of manufacturing parameters and tools during "paste special" and UDF placement procedures. ----- Enable parameters and tool updation dialog.	yes, no	yes
Manufacturing	mfg_tool_preview_by_pick	CHANGED	Enables preview of a milling tool at a specified location on the tool path or geometry of the part.	yes, no	no ----- yes

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Manufacturing	ncmdl_bar_stock_part_path	CHANGED	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<creo2.0_loadpoint>\text\ncmdl_data\nncmdl_bar.prt ----- <creo3.0_loadpoint>\text\ncmdl_data\nncmdl_bar.prt
Manufacturing	ncmdl_billet_stock_part_path	CHANGED	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<creo2.0_loadpoint>\text\ncmdl_data\nncmdl_billet.prt ----- <creo3.0_loadpoint>\text\ncmdl_data\nncmdl_billet.prt
Manufacturing	use_strict_cl_file_names	DELETED	Use strict cl file names.	yes, no	no
Manufacturing	workpiece_instances	CHANGED	Cut out are created in Workpiece but not the family table instances.	none, operations ----- none, operations, steps, operations_and_steps	none
Miscellaneous	conf_mouse_anim	DELETED	Display the presenter's mouse motions when observing a shared Creo Parametric session.	yes, no	no
Model Display	blended_transparency	CHANGED	Yes - Transparent colors will appear using alpha blending (if supported) when the model is shaded.	yes, no, independent	yes ----- independent
Model Display	datum_point_symbol	CHANGED	Specifies the shape of datum point symbols displayed in Part or Assembly mode.	cross, circle, triangle, square ----- dot, cross, circle, triangle, square	cross
Model Display	enable_ambient_occlusion	NEW	Enable ambient occlusion. This simulates the effect of light radiating from a real model by darkening the occluded areas.	yes, no	no
Model Display	max_image_dimension	CHANGED	Set the upper bound dimension on images loaded into Creo Parametric. Images larger than this limit will be rescaled to this upper bound during loading.	4096, 1024, 2048, 8192, 16384	4096 ----- 16384
Model Display	model_detail_options_file	CHANGED	Sets the path to the file which contains the default model detail option values for new models.		<creo2.0_loadpoint>\text\3d_inch.dtl ----- <creo3.0_loadpoint>\text\3d_inch.dtl

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Model Display	smooth_lines	DELETED	Choose whether to smooth lines by default or not.	yes, no	no
Model Display	update_old_appearance	NEW	Update pre-Creo 3.0 appearances to their latest values. Always - update without warning, Never - do not update, Prompt - issue a warning dialog allowing users to choose to update or not.	never, always, prompt	always
Notification Manager	nmgr_affected_by_dma	NEW	Show notifications for models/features that are affected by Creo direct modifications.	yes, no	yes
Notification Manager	nmgr_dont_show_all	NEW	Ignore current configurations and do not display any notifications.	yes, no	no
Notification Manager	nmgr_failed	NEW	Show notifications for failed features/components.	yes, no	yes
Notification Manager	nmgr_misssing_mdl	NEW	Show notifications for models that could not be reterived.	yes, no	yes
Notification Manager	nmgr_modified_in_dma	NEW	Show notifications for models/features that were modified in Creo Direct.	yes, no	yes
Notification Manager	nmgr_outdated_mathcad	NEW	Show notifications for models that contain MathCad Sheet with values different from Creo values.	yes, no	yes
Notification Manager	nmgr_outdated_non-native	NEW	Show notifications for non-native models that are not up-to-date with respect to source.	yes, no	yes
Notification Manager	nmgr_outdated_pdm	NEW	Show notifications for models that are not up-to-date with recent PDM model version.	yes, no	yes
Notification Manager	nmgr_pending_dma	NEW	Models that were modified in Creo Direct and not reviewed. The changes should be accepted or rejected in the View Creo Changes dialog	yes, no	yes
Notification Manager	nmgr_reference_changed	NEW	Show notifications for models/features whose references were changed since last regeneration.	yes, no	yes
Notification Manager	nmgr_reference_out_of_session	NEW	Show notifications for models/features with references not in session.	yes, no	no
Photorender	default_scene_filename	CHANGED	Complete path to the scene file to be used as default scene file.		<creo2.0_loadpoint>\graphic-library\scenes\creo_default_scene.scn ----- <creo3.0_loadpoint>\graphic-library\scenes\creo_default_scene.scn

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Piping	isogen_attribute_map_file	CHANGED	Specifies the mapping between isogen parameter and attribute name.		<creo2.0_loadpoint>\text\piping_data\isodata\isogen_attribute_map.ptd ----- <creo3.0_loadpoint>\text\piping_data\isodata\isogen_attribute_map.ptd
Piping	isogen_endtype_map_file	CHANGED	Specify name of file mapping Pro/Piping endtypes with ISOGEN endtypes		<creo2.0_loadpoint>\text\piping_data\isodata\isogen_endtype_map.ptd ----- <creo3.0_loadpoint>\text\piping_data\isodata\isogen_endtype_map.ptd
Piping	isogen_nominal_size_map_file	CHANGED	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches		<creo2.0_loadpoint>\text\piping_data\isodata\isogen_nominal_size_map.ptd ----- <creo3.0_loadpoint>\text\piping_data\isodata\isogen_nominal_size_map.ptd
Piping	isogen_symbol_map_file	CHANGED	Specify name of file for mapping Pro/Piping selection names with ISOGEN symbol keys (SKEY)s		<creo2.0_loadpoint>\text\piping_data\isodata\isogen_symbol_map.ptd ----- <creo3.0_loadpoint>\text\piping_data\isodata\isogen_symbol_map.ptd
Piping	pipeline_assembly_library_dir	CHANGED	Sets the default pipeline assembly library directory.		<creo2.0_loadpoint>\text\piping_data\pipelinelib\ ----- <creo3.0_loadpoint>\text\piping_data\pipelinelib\

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Piping	pipng_appearance_map_file	CHANGED	Sets the default piping appearance map filename.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_appearance. map ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_appearance. map
Piping	pipng_bolt_nut_select_file	CHANGED	Piping bolts and nuts selection file.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_bolt_nut_sel ect.ptd ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_bolt_nut_sel ect.ptd
Piping	pipng_end_compatibility_file	CHANGED	Sets the default piping end compatibility filename.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_end_compati bility.ptd ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_end_compati bility.ptd
Piping	pipng_fitt_category_map_file	CHANGED	Sets the default piping fitting category map filename.		<creo2.0_loadpoint>\text\piping_data \sample_project\piping_fitt_category _map.ptd ----- <creo3.0_loadpoint>\text\piping_data \sample_project\piping_fitt_category _map.ptd
Piping	pipng_fitt_library_dir	CHANGED	Sets the default piping fitting library directory.		<creo2.0_loadpoint>\text\piping_data \fittinglib\ ----- <creo3.0_loadpoint>\text\piping_data \fittinglib\

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Piping	pipng_insulation_dir_file	CHANGED	Sets the default piping insulation directory filename.		<creo2.0_loadpoint>\text\piping_data \sample_project\piping_insulation_dir .ptd ----- <creo3.0_loadpoint>\text\piping_data \sample_project\piping_insulation_dir .ptd
Piping	pipng_manufacture_dir_file	CHANGED	Sets the default piping manufacture directory filename.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_manufacture _dir.ptd ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_manufacture _dir.ptd
Piping	pipng_material_file	CHANGED	Sets the default piping material filename.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_material.ptd ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_material.ptd
Piping	pipng_mcat_dir	CHANGED	Sets the default piping master catalog directory.		<creo2.0_loadpoint>\text\piping_data \master_catalog\ ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\ 
Piping	pipng_mcat_dir_file	CHANGED	Sets the default piping master catalog directory filename.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_mcat_dir.ptd ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_mcat_dir.ptd
Piping	pipng_project_data_dir	CHANGED	Sets the default Project data directory.		<creo2.0_loadpoint>\text\piping_data \sample_project\ ----- <creo3.0_loadpoint>\text\piping_data \sample_project\ 

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Piping	piping_spec_dir_file	CHANGED	Sets the default piping specification directory filename.		<creo2.0_loadpoint>\text\piping_data \sample_project\piping_spec_dir.ptd ----- <creo3.0_loadpoint>\text\piping_data \sample_project\piping_spec_dir.ptd
Piping	units_system_dic_file	CHANGED	Specifies the file containing the mapping for MCAT filename and units.		<creo2.0_loadpoint>\text\piping_data \master_catalog\piping_units_system _dic_file.ptd ----- <creo3.0_loadpoint>\text\piping_data \master_catalog\piping_units_system _dic_file.ptd
Printing & Plotting	u3d_create_color_surfaces	NEW	YES- Exports color of each surface of model. NO- Exports one color per model. Default NO.	yes, no	no
Reference Control	allow_direct_ref_to_layout_file	NEW	Allow direct references to layout files through layout feature.	yes, no	no
Reference Control	def_comp_ref_ctrl_exception	NEW	Default setting for newly created components that allows/disallows referencing entities forbidden by Reference Control settings.	yes, no	yes
Reference Control	def_multiple_backup_prompt_num	NEW	Number of selected components that will invoke confirmation before multi-component backing up placement references.		100
Reference Control	def_obj_feat_refs_to_backup	NEW	Default setting that controls the level of feature backup for newly created models. Other_models: all references to other models will be backed up, Non_permitted: only references that were created despite the initial restrictions will be backed up.	none, other_models, non_permitted	non_permitted
Reference Control	def_obj_place_refs_to_backup	NEW	Default setting that controls the level of placement backup for newly created models. Other_models: all references to other models will be backed up, Non_permitted: only references that were created despite the initial restrictions will be backed up.	all, none, other_models, non_permitted	non_permitted
Reference Control	def_obj_ref_ctrl_exception	NEW	Default setting for newly created models that allows/disallows referencing entities forbidden by Reference Control settings.	yes, no	yes

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Reference Control	default_comp_invalid_refs	DELETED	Set the specified handling method of out-of-scope Accessible references to newly added components.	copy, prohibit	copy
Reference Control	default_object_invalid_refs	DELETED	Sets default condition for reference handling. Prohibit - System will abort all attempts to create ext. reference that violates scope. Copy - System will issue warning upon all attempts to create ext. reference that violates scope.	copy, prohibit	copy
Reference Control	feat_references_to_backup	NEW	Level of backup for feature references	none, other_models, non_permitted	non_permitted
Reference Control	placement_references_to_backup	NEW	Level of backup for placement references	all, none, other_models, non_permitted	non_permitted
Reference Control	ref_control_allow_exceptions	NEW	Check to allow referencing components that are not permitted by the above setting	yes, no	yes
Reference Control	scope_invalid_refs	DELETED	Prohibit - System will abort all attempts to create external reference violating the scope. Backup - Warning appears. Abort reference creation or declare as out-of-scope reference. If you declare, backup copies to part/assembly and backup is referenced.	copy, prohibit	copy
Sheetmetal	die_models_directory	NEW	Sets path to directory that contains the die models.		
Sketcher	dma_sketcher_guide_width_factor	NEW	Guide width as a proportion of sketched line width		0.75
Sketcher	sketcher_disp_guides	NEW	Shows guides when sketching.	yes, no	yes
Sketcher	sketcher_snap_sensitivity	CHANGED	sketcher snap sensitivity ----- Adjusts the sensitivity of the snapping to geometry.	high, medium, low, very_high, very_low, very_high, very_low ----- high, medium, low, very_high, very_low	very_high ----- very_high
Style Feature	style_merge_surfaces	NEW	Merge surfaces on exit from the Style Feature	yes, no	yes
System	csv_format_delimeter	CHANGED	Define the delimiter character in BOM format file: Comma, TAB, Semicolon, Space or other user-defined. ----- Define the delimeter character in BOM format file: Comma, TAB, Semicolon, Space or other user-defined	comma, tab, semicolon, space	

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
System	web_browser_homepage	CHANGED	Enter location of Creo Parametric browser home page.		<creo2.0_loadpoint>\help\usascii\resource_center_index.html ----- <creo3.0_loadpoint>\help\usascii\resource_center_index.html
System	windows_browser_type	CHANGED	Select the type of embedded browser to use.	ie_in_process, ie_in_sep_process, mozilla_based_browser ----- ie_browser	ie_in_process ----- ie_browser
Update Control	child_of_layout_backup_update	NEW	Set the default update state for a child referencing the layout feature's geometry.	manual, auto, notify	auto
Update Control	copied_feat_refs_update_state	NEW	Copied Feature References Update	manual, auto, notify, no_dependency	auto
Update Control	copied_place_refs_update_state	NEW	Copied Placement References Update	manual, auto, notify, no_dependency	auto
Update Control	direct_child_of_layout_update	NEW	Set the default update state for a child directly referencing the layout model through layout feature.	manual, auto, notify	notify
Update Control	individual_child_update	NEW	Allow individual update control for children of layout feature.	yes, no	yes
Update Control	new_cg_update_state	NEW	Set the default update state for newly created Copy Geometry features relative to the reference model.	manual, auto, notify	auto
Update Control	new_layout_feature_update_state	NEW	Define the default update state of a new DSF created.	manual, auto, notify	notify
User Interface	drm_policy_server	DELETED	Allows a user to preset the server policy that is to be used when Apply Policy command is utilized.		
User Interface	initial_drm_warning	DELETED	Warns the user that they are initiating a DRM session. When set to "no", this warning dialog will be skipped.	yes, no	yes
Without category	accessory_window_above	CHANGED	Yes - Displays and keep the accessory window above the active window. No - Does not keep the accessory window above the active window. ----- Yes - Displays and keeps the accessory window above the active window. No - Does not keep the accessory window above the active window.	yes, no	no

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Without category	auto_center_dimension	NEW	Automatically centers the dimension text between witness lines. For common reference dimensions, controls the centering of the first dimension only.	yes, no	no
Without category	backup_reference_information	NEW	Yes-Backup the reference information of a feature once the feature regenerates successfully. No- Do not save the reference information of a feature.	yes, no	yes
Without category	comp_placement_legacy_move	NEW	Show the legacy Move tab in the Component Placement dashboard	yes, no	yes
Without category	confirm_on_edit_foreign_models	NEW	Provides a confirmation dialog when attempting to make design changes to foreign models	yes, no	no
Without category	des_exp_folder	NEW	Sets the directory for the Design Exploration session. The default location is in a default system folder. Type a valid file path to change the directory to a user defined one.		
Without category	edge_tess_quality	CHANGED	Set the accuracy level for arc tessellation. ----- Set the accuracy level for arc tessellation.	high, medium, low	low
Without category	flex_display_dflt_bounding_edges	NEW	Turns on/off highlighted display of default bounding edges in flexible modeling features that can have multiple solutions.	yes, no	yes
Without category	flex_dma_reduce_solution_number	NEW	Reduces the number of default solutions found in Flex features, when modifying geometry with mixed convexity contours.	yes, no	yes
Without category	flex_tan_propagation_default	NEW	Defines default state of the Tangency Propagation option in Flexible Modeling features.	off, on	off
Without category	intf_nx_version	NEW	Sets the version of NX interface.	6, 7, 8	6
Without category	intf3d_out_nx_version	NEW	Allows the user to switch between the default and alternate converter version for export to NX	3, 5, 7	3
Without category	layout_smart_constraints	CHANGED	----- To improve the user experience, specific entity types are appropriately constrained by default (i.e. - fillets with tangency constraints). Yes - specific entity types will be created with appropriate constraints by default; No - constraints will never be automatically created for any entity type.	yes, no	yes

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Without category	measure_ref_option_status	CHANGED	Determine AsObject (AsCenter et al.) checkbox option behavior in Measure Tool ----- Determine As Object (As Center et al.) checkbox option behavior in Measure Tool	default, all_on, all_off, keep_last	default
Without category	pro_design_test_mode_dir	DELETED	Sets the default directory for the Design Test mode data. If not specified, your current working directory is the default. Use the full pathname to avoid problems.		
Without category	ptc_manikin_library_path	CHANGED	Sets the default directory for the Creo Parametric manikin library (or custom manikin libraries you built containing the proper index menu files). Use the full path name to avoid problems. You must restart Creo Parametric for the change to take effect.		<creo2.0_loadpoint>\manikin_lite\ ----- <creo3.0_loadpoint>\manikin_lite\
Without category	relations_ang_eval_diff_check	NEW	Check for differences in the evaluation of relation sets evaluated and saved in versions of Creo 2.0 prior to M110. Warn_message Check and give a warning in the message area. Warn_dialog Check and give a warning in a message window. Ignore Skip the check and warning.	ignore, warn_message, warn_dialog	warn_message
Without category	show_axes_by_view_scope	CHANGED	Determines the scope of the Show Model Annotations tool when showing model datums and selecting a drawing view. For large assembly drawings, use top_model_only to improve performance by restricting the set of axes to be shown.	top_model_only, all_sub_models	all_sub_models ----- top_model_only
Without category	suppress_dim_ghost_with_ctrl	NEW	Suppress the ghost image of dimension when CTRL is pressed. It allows better visibility while selecting further dimension references.	yes, no	no
Without category	tool_mgr_save_pocket_in_xml	CHANGED	Determines whether tool position should be saved in a XML file (*.txf) created in tool manager dialog. ----- Determines whether the tool position should be saved in a XML file (*.txf) created in tool manager dialog.	yes, no	no ----- yes

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Without category	visible_recent_files	CHANGED	Number of visible recent files. The possible range is 1 - 25. ----- Number of visible recent files. The possible range is 1 - 30.		13 ----- 21
Without category ----- Assembly	new_wf5_simp_rep_ui	CHANGED	Yes - Use Component Chooser base Simp Rep UI, No - Use old Simp Rep definition UI.	yes, no	yes
Without category ----- Data Exchange	ecad_load_filtered_holes_to_ui	CHANGED	Controls whether "ECAD import holes configs" (ecad_pin_hole_import & ecad_via_hole_import) will remove holes completely from the import (default) or just pre-set the holes filter UI condition.	yes, no	no
Without category ----- Data Exchange	ecad_outline_holes_exp_method	CHANGED	Select how holes that are placed on the board outline be exported.	default, arcs	default
Without category ----- Data Exchange	intf_pdf_u3d_create_u3d_files	CHANGED	YES - When exporting a Creo Parametric model as PDF U3D file, create a U3D file of the same name as the PDF file.	yes, no	no
Without category ----- Data Exchange	intf2d_out_blanked_layers	CHANGED	Enables export of blanked layer entities. If an entity is placed on a layer that is blanked and this option is set to YES, the entity will be exported.	yes, no	no
Without category ----- Data Exchange	intf3d_out_jt_config_name	CHANGED	Name of JT configuration file.		
Without category ----- Data Exchange	intf3d_out_jt_config_path	CHANGED	Path to JT configuration file.		
Without category ----- Drawing	warn_if_iso_tol_missing	CHANGED	Yes - Returns the user to the dimension properties dialog to select a different table. No - Applies the tolerance according to existing functionality.	yes, no	no
Without category ----- Electromechanical	boeing_extract_bundle	CHANGED	Yes - Uses the extract bundle algorithm. No - Does not use the extract bundle algorithm.	yes, no	no
Without category ----- Electromechanical	ignore_bundles_in_find_cables	CHANGED	Include bundles and overbraids in the query results when the model or logical data is queried for unrouted items.	yes, no	no

## Creo 2.0 to PTC Creo 3.0 Parametric Configuration Options Changes

Category	Name	Status	Description	Values	Default Value
Without category ----- Electromechanical	smaller_bundle_min_bend_r ad	CHANGED	Setup whether to consider the thickness of the bundle when set up the minimum bend radius.	yes, no	no
Without category ----- Electromechanical	trace_thru_splice_connector	CHANGED	Report the from/to connectors of cables passing through splices correctly.	yes, no	no
Without category ----- Layers	save_hidden_items_w_statu s	CHANGED	The items on the "Hidden Items" temporary layer are stored permanently when layer display status is saved with Save Status command.	yes, no	yes
Without category ----- Piping	pipe_3d_bend_theor_int_pts	CHANGED	Yes - Displays theoretical intersections and bends in three-dimensional models.	yes, no	no
Without category ----- Reference Control	default_ext_ref_scope	CHANGED	Set default scope for externally referenced models. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all, none, subassemblies, skeleton_model	all
Without category ----- System	display_ie8_warning_dialog	CHANGED	Show/Hide the warning dialog related to Internet Explorer 8 running in Compatibility Mode	yes, no	yes