Pro/ENGINEER® Wildfire® 5.0 Configuration Options

January 2010

Parametric Technology Corporation

Table of Contents

Pro/ENGINEER Wildfire 5.0 Configuration Options

Name	Category	Description	Values	Default Value
accessory_window_display	Without category	Docked - Displays the accessory window embedded in the active window. Undocked - Displays the accessory window as a standalone window.	undocked, docked	docked
accuracy_lower_bound	Environment	Enter an accuracy value to override the default lower limit.The minimum value for relative accuracy is 1E-06.		0.0001
acis_export_params	Data Exchange	For ACIS interface allows to export object, feature and entity level parameters.	yes, no	no
acis_export_units	Data Exchange	Allows selection of model units for ACIS export.	in, ft, m, mm, cm, micron, default	default
acis_out_version	Without category	Allows the user to choose ACIS export version.	4, 5	5
acrobat_reader	Printing & Plotting	Sets the Adobe Acrobat Reader command path. This value will be used to launch the reader after exporting a PDF document.		
add_java_class_path	Application Programming Interfaces	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.		
add_lower_level_comps_to_layer	Layers	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
add_weld_mp	Weld	Yes - System includes welds when calculating mass properties and approximates light weight weld mass properties. No - System excludes welds when calculating mass properties.	yes, no	no
advanced_intersection	Assembly	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	no
advanced_style_surface_edit	Without category	Use advanced options by default for Surface Edit in Style.	yes, no	no
ae_propagate_detail_dependency	User Interface	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

Name	Category	Description	Values	Default Value
aec_dwg_anno_attrpar_spec_file	Piping	Specifies the aec drawing annotation attribute/parameter specification file name.		aec_dwg_anno_attrparam_spec.ptd
aec_object_type_file	Piping	Specifies the aec object type file name.		aec_object_type.ptd
aec_parameter_file	Without category	Specifies the file name of the user defined piping parameter codes.		
af_copy_references_flag	User Interface	Toggle display of Copy References column in Annotation Feature definition dialog	no, yes	no
align_cable_bundles	Electromechanical	Align NEWLY CREATED bundles where they meet or branch out	yes, no	yes
allow_anatomic_features	Features	Allow the creation of pre-Pro/E 2000i geometry features including ear, neck, lip, slot, flange, shaft, local push, radius dome and section dome.	yes, no	no
allow_confirm_window	User Interface	Display a confirmation window when exiting Pro/ENGINEER. This enables you to use the mouse to confirm/cancel exiting Pro/ENGINEER.	yes, no	yes
allow_delete_many_in_drawings	Without category	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
allow_float_opt_checkout	System	Yes - Allows checkout failure for floating modules.	no, yes	no
allow_fully_dependent_copy	Features	Enables the usage of advanced copy/paste functionality tocontrol degree of dependency on the pasted object.	yes, no	yes
allow_harn_mfg_assy_retrieval	Electromechanical	Allow to retrieve a harness mfg assembly directly.	yes, no	no
allow_import_file_extension	Data Management	Allow additonal files with extensions to be shown in the import dialog		
allow_move_attach_in_dtl_move	Drawing	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
allow_move_view_with_move	Drawing	Disallow the movement of drawing views with the mouse.	yes, no	no
allow_package_children	Assembly	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

Name	Category	Description	Values	Default Value
allow_ply_cross_section	Miscellaneous	Yes - Enables Pro/COMPOSITE to create a cross-section of composite plys.	yes, no	yes
allow_redo_intersections	Assembly	Yes - Displays the ReIntersect command in the ASSY FEAT menu.	yes, no	no
allow_ref_scope_change	Without category	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
allow_refs_to_geom_reps_in_drws	Drawing	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
allow_rfs_default_gtols_always	Dimensions & Tolerances	Yes - Enables RFS/Default gtols to be created even when not permitted by ANSI standard.	no, yes	no
allow_save_as_instance	Assembly	Allows the option to save a copy of the active model as a family table instance.	yes, no	no
allow_save_failed_model	Assembly	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
allow_shrink_dim_before	Casting & Mold Design	Allow calculation before relations.	yes, no	no
allow_solid_perspective_hlr	Model Display	Yes - Allows a solid to be displayed with hidden lines while in perspective.	no, yes	yes
allow_udf_style_cosm_threads	Features	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
allow_update_part_simp_rep_data	Assembly	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
allow_workpiece_silhouette	Without category	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

Name	Category	Description	Values	Default Value
analysis_crt_tmp_file_on_regen	File Storage & Retrieval	Determines whether regeneration of analysis features creates/updates tmp files	yes, no	yes
ang_dim_in_screen	Dimensions & Tolerances	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
ang_units	Environment	Sets the display of angular dimensions to decimal degrees (ang_deg), degrees and decimal minutes (ang_min) or degrees, minutes and decimal seconds (ang_sec).	ang_deg, ang_min, ang_sec	ang_deg
angle_grid_interval	Features	Modify grid space in angular units for handle movement.		1
angular_tol	Dimensions & Tolerances	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
angular_tol_0.0	Dimensions & Tolerances	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
angular_tol_0.00	Dimensions & Tolerances	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
angular_tol_0.000	Dimensions & Tolerances	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
angular_tol_0.0000	Dimensions & Tolerances	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
angular_tol_0.00000	Dimensions & Tolerances	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
angular_tol_0.000000	Dimensions & Tolerances	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
animate_explode_states	Model Display	Animate the transition between explode states and mold opening steps.	yes, no	yes

Name	Category	Description	Values	Default Value
arc_radius_limit	Without category	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
ask_designate_owners	Without category	Show the both required and optional owners to be designated.	yes, no	yes
assemble_parent_connector	Electromechanical	Assemble parent connectors on locations referencing a subconnector.	yes, no	yes
assy_mfg_open_mode	Manufacturing	No Default Application set for opening Assembly NC Models.	mfg, process, feature	
atb_auto_check_on_activate	Data Exchange	Sets state of ATB 'Auto Check Status On Activate' toggle.	on, off	off
atb_auto_check_on_retrieve	Data Exchange	Sets state of ATB 'Auto Check Status On Retrieve' toggle.	on, off	on
atb_auto_check_on_update	Data Exchange	Sets state of ATB 'Auto Check Status On Update' toggle.	on, off	off
atb_ident_cadds_files	Data Exchange	Allows to identify CADDS files in the file open browser.	yes, no	no
atb_prod_asm_upd_by_comp_ids	Data Exchange	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
atb_search_path	Without category	Extends search path for ATB Update and ATB Check Status commands.		
atb_show_log	Data Exchange	Sets state of ATB 'Show Log' toggle.	on, off	on
attach_menumanager	Without category	Setup menu manager attachment to the main Pro/E window.	yes, no	no
auto_ae_param_file	User Interface	Define path and file name of external file used to define parameters that are automatically created in all Annotation Elements.		
auto_assembly_with_layouts	Assembly	Yes - Enables automatic assembly. No - Disables automatic assembly.	yes, no	yes
auto_associate_dimensions	Data Exchange	Yes - The drawing setup option associative_dimensioning is also set to Yes. The system tries to associate imported IGES dimensions (not yet associative) with corresponding imported geometry.	no, yes	no
auto_backup_new_placemnt_refs	Assembly	Yes - Create Component Reference Backup, No - Do not Create Component Reference Backup.	yes, no	no

Name	Category	Description	Values	Default Value
auto_constr_always_use_offset	Assembly	Control whether auto constraint should create offsets. Yes - Auto constraint alw ays creates offsets. No - Auto constraint snaps align or mate if surfaces are wi thin tolerance. Never - Auto constraint never create offsets.	yes, no, never	no
auto_constr_offset_tolerance	Drawing	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
auto_convert_cables	Electromechanical	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
auto_evaluate_simprep_rules	Assembly	If the option is "yes" newly created rep would be updated each time on retrieve/regeneration.	no, yes	no
auto_include_annot_plane	User Interface	Control the default setting of the check box in the Propagate annotation dialog box which controls whether to pass all annotations plane references.	no, yes	yes
auto_place_max_number	Assembly	Set the default value for the maximum number of matches allowed.		5
auto_propagate_ae	User Interface	Yes - causes the automatic, local propagation of AE's after a supported feature has been created.	no, yes	no
auto_regen_views	Drawing	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
auto_show_3d_detail_items	Without category	When set to yes, AEs parallel to the view will be displayed while creating a new view.	yes, no	yes
auto_traceback	System	When set to yes, traceback.log file will be generated when fatal error occurs.	yes, no	no
auto_update_intersected_models	Assembly	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	yes
auto_xml_on_retrieve	Electromechanical	Automatically load Cabling Logical Reference from an XML file when retrieving the cabling assembly	yes, no	no
auto_xml_on_save	Electromechanical	Automatically create an XML Logical Reference when saving a cabling assembly	yes, no	no
autobuildz_enabled	Drawing	Loads AutobuildZ application.	yes, no	no

Name	Category	Description	Values	Default Value
autodrill_udf_dir	Manufacturing	Storage directory for holemaking MUDF's that are used by Pro/NC Auto-drilling functionality		
autodrilling_config_filename	Manufacturing	Retrieve automatically and load the configuration file for the auto-drilling table.		autodrilling
autodrilling_holestyle_param	Manufacturing	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.		
autohide_copied_group_af	Features	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
autohide_external_sketch	Features	If set to yes, a Sketch feature is automatically hidden after being used as external sketch for a sketch based feature.	yes, no	yes
autoplace_single_comp	Assembly	Assembling components with an interface will automatically place the component at the first position satisfying the interface definitions.	no, yes	yes
autoround_max_n_chains_per_feat	Features	Determines the maximum number of edge chains that the Auto-Round algorithm will attempt to round in each Auto-Round member		20
autoroute_path_param_name	Electromechanical	Set the wire parameter to use to select a path during autorouting.		use_path
auxapp_popup_menu_info	User Interface	Enable display of popup menu info in the trail file.	yes, no	no
bell	Environment	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
bitmap_size	Data Management	Works in conjunction with save_bitmap. Determines the size (squared) of the bitmap image that is stored.		200
blank_layer	Layers	Blanks specified layers when you begin a Pro/ENGINEER session. The value is the layer ID.		
blended_transparency	Model Display	Yes - Transparent colors will appear using alpha blending (if supported) when the model is shaded.	no, yes	yes
bm_graph_tool	User Interface	BMX graph tool	default, excel_linked, excel embedded	default
bmgr_pref_file	User Interface	Specifies the location of the Graph Tool preference file.		

Name	Category	Description	Values	Default Value
bmx_param_restrictions	Without category		yes, no	yes
boeing_extract_bundle	Without category	Yes - Uses the extract bundle algorithm. No - Does not use the extract bundle algorithm.	yes, no	no
bom_format	Drawing	Sets the BOM format file to be used for a customized BOM.Specify name and path.		
browser_favorite	File Storage & Retrieval	Specifies the directories visible in the file browser for quick navigation.Use the full path to avoid problems.		
bump_revnum_on_retr_regen	Assembly	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
button_name_in_help	User Interface	Yes - The name and menu of any selected menu option displays, in English, in the help text associated with that button.	yes, no	no
cable_int_portions_for_clr	Electromechanical	No - Global clearance checking for internal cable portions is excluded. Yes - Global clearance checking for internal cable portions is included.	no, yes	no
cables_with_hlr	Electromechanical	Cables are not hidden behind objects.	no, yes	yes
cadds_import_layer	Data Exchange	Allows import of CADDS5 layering.	yes, no	yes
campost_dir	Manufacturing	Specifies the CAM-Post load point directory to enable you to directly set-up the post-processors and generate MCD files.		
can_snap_to_missing_ref	Assembly	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
capped_clip	Model Display	Yes - Displays the model as a solid when shaded and clipped. No - Displays the model as surfaces when shaded and clipped.	no, yes	yes
catia_out_to_existing_model	Data Exchange	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
centimeter_grid_interval	Features	Modify grid space in centimeter units for handle movement.		0.1

Name	Category	Description	Values	Default Value
cgm_use_pentable	Data Exchange	No - Do not use a defined pen table for formatting of CGM export.	no, yes	no
chamfer_45deg_dim_text	Drawing	Controls the display of chamfer dimension text without affecting the leader. This only affects the text of newly created dimensions. ASME/ANSI is the default.	asme/ansi, iso/din, jis	asme/ansi
check_interface_criteria	Assembly	Check for interfaces when placing a component.	yes, no	no
check_same_assy_parts	Assembly	Determines whether to permit different names for different part occurrences, or not in mirror subassembly UI.	yes, no	yes
check_udf_int_key	Without category	Yes - updates UDF internal keys upon group's retrieval. No - skips keys update.	yes, no	yes
chk_part_surfs_profpock	Manufacturing	Yes - Includes all reference part surfaces as check surfaces for profiling and pocketing NC sequences.	yes, no	no
cl_arrow_scale	Manufacturing	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
cl_rotate_sign_convention	Manufacturing	TOOL or TABLE - defines ROTATE sign direction convention based with respect to either rotation table or cutting tool.	table, tool	table
clearance_triangulation	Environment	Specifies the quality of surface triangulation used to obtain initial guess points for clearance and interference calculations.	none, low, medium, high	none
clip_always	Model Display	Yes - Display clipping after Visibilities dialog box is closed.	no, yes	yes
clock	Environment	Yes - Displays the clock while Pro/ENGINEER is working.	yes, no	no
clr_print_plus_minus	Environment	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
cmm_custom_tool_param_file	Manufacturing	Defines the file that contains the user-defined tool parameters' list.		
collect_dims_in_active_layer	Without category	This config options allows dimensions to be collected on the active layer.	yes, no	yes
color	Colors	Turns the colors on or off. Turning colors off displays the model wireframe in white.	no, yes	yes

Name	Category	Description	Values	Default Value
color_ramp_size	Colors	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
color_resolution	Colors	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
color_windows	Colors	All_windows - Wireframe colors display in the main window and all auxiliary windows. One_window - Colors display only in the main window. Auxiliary windows display wireframe in the default color.	all_windows, one_window	all_windows
combined_views_customization	Without category	Enable / Disable combined views customization.	yes, no	yes
comp_angle_offset_eps	Assembly	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
comp_assemble_start	Assembly	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 thendefine placement constraints.	default, package, constrain_in_window, move_then_place	
comp_assemble_with_interface	Without category	Settings for assembling component with interface(s).	default_multi, default_single, from_list, none	default_multi
comp_interface_placement	Without category	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
comp_placement_assumptions	Assembly	Use placement assumptions when placing components.	yes, no	yes
comp_rollback_on_redef	Assembly	No - Assembly is not rolled back when the user redefines a component.	yes, no	yes

Name	Category	Description	Values	Default Value
comp_snap_angle_tolerance	Without category	Specify the angle tolerance for snapping while freeform dragging components. Default value is 30.0 degrees.		30
comp_snap_dist_tolerance	Without category	Specify the distance tolerance for snapping while freeform dragging components.Default value is 0.1 (relative to size of component being assembled).		0.1
compact_cables_packing	Electromechanical	Setup whether the new packing of cables should be applied.	yes, no	no
company_name	Environment	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.		
compress_output_files	File Storage & Retrieval	Yes - Stores object files in a compressed format.Note: Compressed files are usually smaller, buy may be larger,than uncompressed files. Compressed files are slower to read and to write to.	yes, no	no
conf_mouse_anim	Miscellaneous	Display the presenter's mouse motions when observing a shared Pro/ENGINEER session.	no, yes	yes
contact_surfaces_gap	Without category	Sets the maximum allowable gap between weld contact surfaces.		0
convert_to_wf3_replace	Without category	Convert to Wildfire3 replace functionality.	yes, no	no
copy_dxf_dim_pict	Data Exchange	YES - Imports each component of AutoCAD dimension separately. NO - Imports AutoCAD dimensions as dimensions in Pro/ENGINEER. AS_SYMBOL - Imports AutoCAD dimensions as symbols in Pro/ENGINEER.	yes, no, as_symbol	no
copy_geom_update_pre_2000i_dep	Assembly	Yes - Flags independent copy geom features in a pre-2000i model as modified when retrieved into Pro/ENGINEER. Save the model immediately to update the model's copy geom dependency information.	no, yes	
copy_geometry_method	Assembly	Sets the default copy geometry reference(s) type	publish_geometry, references	publish_geometry
copy_set_datum_attr	Features	If set to yes, allow to copy set datum associated with datum features..	yes, no	yes
cpus_to_use	System	Set up the number of threads including main thread		2

Name	Category	Description	Values	Default Value
create_drawing_dims_only	Drawing	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
create_fraction_dim	Dimensions & Tolerances	Yes - All dimensions created will be displayed as fractions.	yes, no	no
create_numbered_layers	Layers	Yes - Creates default layers named 1 through 32.	yes, no	no
create_temp_interfaces	Assembly	Allow the automatic creation of interfaces based on previous assembly instructions.	yes, no	no
cri_grafting_enable	Data Exchange	Activates "Graft features" button under File menu. This allows user to graft features created in CRI model to active model.	yes, no	no
cti_class_color_file	Electromechanical	Sets the CTI class color file for CTI class color displayduring electrical clearance and creepage analysis.Use the full pathname to avoid problems.		
curr_proc_comp_def_color	Assembly Process	Specifies the default color used to display the current component in a process assembly.		0
curr_proc_comp_def_font	Assembly Process	Specifies the default font used on the current component in a process assembly.		
custom_rainbow	Mechanica	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.		
datum_point_symbol	Model Display	Specifies the shape of datum point symbols displayed in Part or Assembly mode.	dot, cross, circle, triangle, square	cross
dazix_default_placement_unit	Data Exchange	Specifies the units to be used for data imported in Dazix files.	micron, mm, thou	
dazix_export_mounthole	Data Exchange	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
dazix_z_translation	Data Exchange	Yes - Passes the objects in the .edn files through z translation.	yes, no	yes

Name	Category	Description	Values	Default Value
def_layer	Layers	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_3d_items, layer_ae_driving_dim, layer_all_detail_items, layer_annotation_element, layer_assem_member, layer_assy_cut_feat, layer_axis, layer_axis_ent, 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	

Name	Category	Description	Values	Default Value
def_layer	Layers		layer_cosm_sketch, layer_csys, layer_csys_ent, 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	

Name	Category	Description	Values	Default Value
def_layer	Layers		layer_draft_others, layer_draft_refdim, layer_draft_xsec, layer_driven_dim, layer_dwg_table, layer_ext_copy_geom_feat, layer_feature, layer_geom_feat, layer_gtol, layer_hole_feat, layer_hole_thread, layer_intchg_funct, layer_intchg_simp, layer_nogeom_feat, layer_note, layer_parameter_dim, layer_part_refdim, layer_point	
default_abs_accuracy	Environment	Defines the default absolute part accuracy.		0
default_ang_dec_places	Drawing	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
default_ang_units	Dimensions & Tolerances	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
default_comp_geom_scope	Without category	Set thespecified Shared Geometry reference control conditions to newly added components.	all, pg, none	all
default_comp_invalid_refs	Without category	Set the specified handling method of out-of-scope Accessible references to newly added components.	copy, prohibit	copy

Name	Category	Description	Values	Default Value
default_comp_scope_setting	Without category	Set the specified Accessible reference control conditions to newly added components.	all, none, subassemblies, skeleton_model	all
default_comp_skel_geom_scope	Without category	Set the specified Shared Geometry reference control conditions to newly added Skeleton components.	all, pg, none	all
default_comp_skel_scope_setting	Without category	Set the specified Accessible reference control conditions to newly added Skeleton components.	all, none, subassemblies, skeleton_model	all
default_dec_places	Dimensions & Tolerances	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
default_dep_copy_option	Features	Sets Full Dependent Copy as a default copy option.	full_dep, dep_dim	dep_dim
default_dim_num_digits_changes	Dimensions & Tolerances	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
default_draw_scale	Drawing	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
default_ext_ref_scope	Without category	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
default_font	User Interface	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.		
default_font_kerning_in_drawing	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
default_geom_analysis_type	Without category	Default save type for geometry analysis.	quick, saved, feature	quick

Name	Category	Description	Values	Default Value
default_geom_scope	Without category	Default value for Geometry Scope allowed for referencing.	all, pg_if_exists, pg_only	all
default_layer_model	Layers	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.		
default_mold_base_vendor	Casting & Mold Design	Default Value for Mold Base vendor.	futaba_mm, dme, hasco, dme_mm, hasco_mm	
default_obj_constraints_scope	Without category	Set the specified Shared Placement reference control conditions to newly created models if template models were not used.	all, interfaces, none	all
default_object_geom_scope	Without category	Set the specified Shared Geometry reference control conditions to newly created models if template models were not used.	all, pg, none	all
default_object_invalid_refs	Without category	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
default_object_scope_setting	Without category	Set default condition for reference control. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly ang higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all, none, subassemblies, skeleton_model	all
default_placement_scope	Without category	Default value for Component Placement Reference filter.	all, intrf_if_exists, intrf_only	all
default_placement_surfacefinish	Dimensions & Tolerances	Sets the default placement type for surface finishes within the surface finish UI.	with_leaders, on_entity, normal_to_entity, free	normal_to_entity
default_ramp_size	Model Display	Specifies a number of shades of gray to show shading of non-colored objects.		
default_scene_filename	Photorender	Complete path to the scene file to be used as default scene file.		<proe_WF_5.0_loadpoint>/graphic-library/scenes//default-enhanced-realism.scn

Name	Category	Description	Values	Default Value
default_shrink_formula	Casting & Mold Design	Default option of shrinkage formula: 1/(1-S) - ASME standard or 1+S (simple).	asme, simple	simple
delete_after_plotting	Printing & Plotting	Yes - A plot file is automatically deleted from the directory in which it resides when it has been successfully plotted.	yes, no	
depthcue_always	Model Display	Depth cue aids in visualizing a wireframe model. With depth cue enabled, wireframe lines become darker as they extend into the screen and lighter as they extend towards the screen. Depth cue is supported only by the appropriate graphics hardware.	no, yes	yes
depthcue_value	Model Display	Aids in visualizing a wireframe model. With depth cue enabled, this option specifies the default percentage of depthcue used. Depth cue is supported only by the appropriate graphics hardware.		
diagram_export_allow	Electromechanical	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.	no, yes	no
diagram_pin_edit_multi_line	Electromechanical	(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 ENDDEF statements for each pin. "no" - The system uses columnar format and	yes, no	no
dialog_translation	User Interface	Specifies the language in which the dialog display when running non-English version of Pro/ENGINEER.	yes, no	yes
dim_fraction_denominator	Dimensions & Tolerances	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
dim_fraction_format	Dimensions & Tolerances	Std - Displays fractional dimensions in the standard Pro/ENGINEER 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

Name	Category	Description	Values	Default Value
dim_inactive_components	Assembly	Display inactive assembly components with stippled transparency and default color (grey). Default value is shade_only.	always, never, shaded_only	shaded_only
dim_offscreen_limit	Dimensions & Tolerances	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
din_std_line_priority	Printing & Plotting	If set to YES line priority in plotting or export is determined according to DIN standard	yes, no	no
dir_cache_max	File Storage & Retrieval	Specify the maximum number of directories to cache.Larger values can speed up file access, but use more memory.		256
direct_vrml	Data Exchange	Yes - Enables direct VRML conversion.	yes, no	no
disable_search_path_check	File Storage & Retrieval	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
disp_in_open_rep_create	Without category	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, default rep, no	no
disp_trimetric_dwg_mode_view	Drawing	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
display	Model Display	Wireframe - All solid edges in white. Hiddenvis - Hidden lines in grey. Hiddeninvis - Hidden lines removed. Shade - Display shaded surfaces. Settings are reflected in printing and some export modes.	wireframe, hiddenvis, hiddeninvis, shade	shade
display_annotations	Environment	Determines whether or not annotations are displayed in 3D models.	yes, no	yes
display_axes	Model Display	Displays or hides datum axes.	yes, no	yes
display_axis_tags	Model Display	Displays or hides datum axis name tags.	yes, no	yes

Name	Category	Description	Values	Default Value
display_comps_to_assemble	Assembly Process	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
display_coord_sys	Model Display	Yes - Coordinate systems will be displayed. No - Coordinate systems will not be displayed.	yes, no	yes
display_coord_sys_tags	Model Display	Yes - Coordinate system name tags will be displayed. No - Coordinate system name tags will not be displayed.	yes, no	yes
display_dwg_sketch_constraint	Drawing	When set to yes, parametric sketching constraints, such as V for vertical, are displayed when a drawing object is selected	yes, no	no
display_dwg_tol_tags	Drawing	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
display_florin	Model Display	Yes: display of the florin (international currency symbol) where possible. No: use the euro symbol exclusively.	no, yes	no
display_full_object_path	Environment	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.	no, yes	no
display_ie8_warning_dialog	Without category	Show/Hide the warning dialog related to Internet Explorer 8 running in Compatibility Mode	yes, no	yes
display_in_adding_view	Drawing	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.	wireframe, minimal_wireframe, default	wireframe
display_internal_cable_portion	Electromechanical	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

Name	Category	Description	Values	Default Value
display_layer	Layers	This causes the specified layers to display when you begin a Pro/ENGINEER session. The variable "value" is the layer Id.		
display_mfg_icon_for_mfg_assy	Manufacturing	Yes- Display manufacturing icon for manufacturing assembly files in Open/Save dialogs.No- the manufacturing assembly files have assembly icons.	yes, no	no
display_pdm_server_quick_view	Without category	Yes - display summary view of PDM information for items in session. It applies to ProductPoint only.	yes, no	yes
display_plane_tags	Model Display	Yes - Datum plane name tags will be displayed. No - Datum plane name tags will not be displayed.	yes, no	yes
display_planes	Model Display	Yes - Datum planes will be displayed. No - Datum planes will not be displayed.	yes, no	yes
display_point_tags	Model Display	Yes - Datum point tags will be displayed. No - Datum point tags will not be displayed.	yes, no	yes
display_points	Model Display	Yes - Datum points and their names will be displayed. No - Datum points and their names will not be displayed.	yes, no	yes
display_popup_viewer	Without category	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
display_route_cable_preview	Electromechanical	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
display_silhouette_edges	Model Display	Sets the display of silhouette edges for wireframe display only.	no, yes	yes
display_thick_cables	Electromechanical	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
display_thick_insulation	Piping	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
display_thick_pipes	Piping	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

Name	Category	Description	Values	Default Value
display_tol_by_1000	Dimensions & Tolerances	If set to Yes, tolerances for non-angular dimensions will be displayed multiplied by 1000.	yes, no	no
display_z_extension_lines	Model Display	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
dm_auto_open_zip	Data Management	YES - Windchill CAD packages or zip files will open automatically. NO - User selects which file in contents to open.	yes, no	yes
dm_cache_limit	Data Management	Enter amount of disk space (in megabytes) to allocate for local file storage. The change to the existing setting will take effect after Pro/ENGINEER is restarted		
dm_cache_mode	Data Management	Indicate which objects will be written to local cache when objects in Pro/E memory are saved.	none, modified, all	all
dm_checkout_on_the_fly	Data Management	Controls the default action presented to the user when Check Out on the fly dialog comes up	checkout, continue	checkout
dm_enable_simple_import_export	Without category		yes, no	yes
dm_fallback_server_location	Data Management	Specifies the server location where objects will be saved when Pro/ENGINEER does not provide an ability to specify the target save location. This option is applicable for Windchill ProductPoint only.		
dm_http_compression_level	Data Management	Set the data compression factor (0-9, 0= no compression) for data exchange with a Windchill server.		
dm_nc_references	Without category	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
dm_offline_after_event	Data Management	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
dm_offline_options_activated	Data Management	Yes - Synchronize, Download and Upload boxes are checked when going online. No - Only Synchronize box is checked when going online.	yes, no	yes
dm_overwrite_contents_on_update	Data Management	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

Name	Category	Description	Values	Default Value
dm_remember_server	Data Management	Primary server/workspace must be set by user for each Pro/E session.	no, yes	yes
dm_save_as_attachment	Data Management	Controls the default option for Save A Copy when models are saved as in non-Pro/E 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	
dm_search_primary_server	Data Management	Search the primary server for dependencies not found in the workspace.	yes, no	yes
dm_secondary_upload	Data Management	Indicate when modified Pro/E objects on a secondary server will be moved from the local cache to the user's workspace on the server.	automatic, explicit	automatic
dm_synchronize_in_background	Data Management	Yes: Enables workspace synchronization in the background. No (Default): Workspace synchronization will happen in the foreground.	yes, no	no
dm_upload_objects	Data Management	Indicate when modified Pro/E objects will be moved from the local cache to the user workspace on the server.	automatic, explicit	explicit
dm_user_presence	Data Management	Enable user presence indicator and actions.This capability requires a connection with a PDM server and requires an installed and running Microsoft communicator client.	yes, no	yes
dont_show_symbol_nodes	Design Manager	Do not show references derived from relations in Parent/Child viewer.	yes, no	no
draw_models_read_only	Drawing	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
draw_points_in_model_units	Drawing	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_ole_image_dpi	Drawing	Resolution of OLE object images. The range of valid values is 0 to 600 dots per inch.		150

Name	Category	Description	Values	Default Value
drawing_setup_file	Drawing	Sets the default drawing setup file option values for your Pro/ENGINEER 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.		<proe_WF_5.0_loadpoint>/text/prodetail.dtl
drawing_shaded_view_dpi	Drawing	Resolution of exported shaded drawing views.The range of valid values is 0 to 600 dots per inch.		150
drawing_view_origin_csys	Drawing	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	
drm_policy_server	User Interface	Allows a user to preset the server policy that is to be used when Apply Policy command is utilized.		
dwg_export_format	Data Exchange	Provides the ability to choose DWG file version when exporting from Pro/E drawing.	14, 2000, 2004, 2007	2007
dwg_unicode_conversion_language	Drawing	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 Pro/ENGINEER session.	file, session, usascii, english, japanese, french, german, russian, italian, spanish, korean, chinese_tw, chinese_cn, hebrew, greek, turkish, czech, polish, hungarian, slovenian, portuguese, slovak	file
dxf_block_to_pro_symbol	Data Exchange	Specifies whether or not DXF BLOCKS to be imported as Pro/E symbols. Yes - Imports BLOCKS as symbol definitions, No - Imports BLOCKS as separate entities.	yes, no	no
dxf_export_format	Data Exchange	Provides the ability to choose DXF file version when exporting from Pro/E drawing.	12, 13, 14, 2000, 2004, 2007	2007
dxf_in_faceted_brep	Data Exchange	Default, faceted boundary representation is allowed in import.	yes, no	yes
dxf_in_proxy_entity	Data Exchange	Controls the input support for ACAD_PROXY_ENTITY.	no, yes	yes

Name	Category	Description	Values	Default Value
dxf_out_comments	Data Exchange	YES - comments (999 group codes) will be created in DXF file exported from Pro/ENGINEER. NO - DXF file will be created without comment lines.	yes, no	yes
dxf_out_drawing_scale	Data Exchange	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
dxf_out_scale_views	Data Exchange	Specify whether or not to scale drawing views when exporting to theDXF or DWG file.Yes - rescales the entire drawing in such a way that the main viewscale becomes 1:1. No - exports without rescaling.	yes, no	no
dxf_out_sep_dim_w_breaks	Data Exchange	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
dxfio_in_sjis	Data Exchange	Determines if the SJIS is supported when importing/exporting DXF files in Japanese	yes, no	yes
dynamic_edit_show_dims	Without category	Controls the display of dimensions during dynamic edit.	yes, no	yes
dynamic_edit_show_sketch_dims	Without category	Controls the display of sketch dimensions during dynamic edit.	yes, no	yes
ecad_area_default_import	Data Exchange	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
ecad_board_csys_def_name	Data Exchange	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.		
ecad_comp_csys_def_name	Data Exchange	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.		
ecad_comp_layer_map	Data Exchange	Allow layer mapping for ECAD component's import into Assembly.	yes, no	yes

Name	Category	Description	Values	Default Value
ecad_comp_xsec_def_name	Data Exchange	Will allow pre-setting default X-section to be used for component outline output to *.eda format.		
ecad_create_hint_add	Data Exchange	Assists in creation of an ecad_hint.map file. Yes - Automatically renames components, if necessary, each time library of component outlines is imported to Pro/ENGINEER. This does not create an ecad_hint.map file. It controls if ecad_hint.add is created.	no, yes	yes
ecad_default_comp_height	Data Exchange	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
ecad_default_comp_place_status	Data Exchange	Sets default component placement status for export (parameter setting will override the default).	unplaced, fixed, placed, mcad, ecad	placed
ecad_exp_both_two_areas	Data Exchange	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
ecad_export_cuts_as_holes	Data Exchange	Exports Pro/ENGINEER cuts as holes to ECAD systems.	yes, no	yes
ecad_export_holes_as_cuts	Data Exchange	Exports Pro/ENGINEER holes as cuts to ECAD systems.	yes, no	no
ecad_import_holes_as_features	Data Exchange	Import sections specified as DRILLED_HOLE as through-all holes. Boards created with Pro/ENGINEER 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
ecad_import_relative_accuracy	Data Exchange	Defines the relative accuracy used in ECAD import.		0.0012
ecad_load_filtered_holes_to_ui	Without category	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
ecad_mapping_file	Data Exchange	Specify the ecad_hint.map file that will be used for ECAD operations.		

Name	Category	Description	Values	Default Value
ecad_missing_component_status	Data Exchange	Setup default status for missing components in "Investigate Placement" Dialog.	keep_missing, delete_missing	keep_missing
ecad_mtg_hole_import	Data Exchange	Setup the default for the import of ECAD Holes type MTG (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
ecad_other_outl_csys_def_name	Data Exchange	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.		
ecad_outline_holes_exp_method	Without category	Select how holes that are placed on the board outline be exported.	default, arcs	default
ecad_panel_csys_def_name	Data Exchange	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.		
ecad_pin_hole_import	Data Exchange	Set up the default for the import of ECAD Holes from type PIN (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	no
ecad_tool_hole_import	Data Exchange	Setup the default for the import of ECAD Holes type TOOL (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
ecad_via_hole_import	Data Exchange	Set up the default for the import of ECAD Holes from type VIA (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
edge_display_quality	Model Display	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.	normal, high, very_high, low	normal
efx_enabled	Data Exchange	Loads EFX application.	yes, no	no
enable_3dmodelspace_browser_tab	Without category	Yes - Display 3D ModelSpace tab in the browser when Pro/ENGINEER starts.No - Display 3D ModelSpace tab only when you click a 3D ModelSpace link in the navigator.	yes, no	yes
enable_absolute_accuracy	Environment	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
enable_acis_export_extension	Data Exchange	Enable ACIS export extension.	yes, no	no

Name	Category	Description	Values	Default Value
enable_advance_collision	Assembly	Enables advance collision settings usage.	yes, no	no
enable_assembly_accuracy	Assembly	No - Disables accuracy modification for assembly objects.	yes, no	yes
enable_cadra_export	Data Exchange	Enables the CADRA command in the EXPORT menu. This enables the creation of a CADRA specific IGES file.	no, yes	no
enable_face_preselection	Without category	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
enable_hlr_hashing	Model Display	Enables hlr hashing	no, yes	no
enable_implied_joints	Assembly	Allow underconstrained components as mechanism connections.	yes, no	yes
enable_offset_fan_curve	Without category	Enables fan curve option for offset tool.	yes, no	no
enable_popup_help	User Interface	Controls pop-up help in dialog boxes. Yes - Help enabled. No - Help disabled.	yes, no	yes
enable_protk_xml_schema	Application Programming Interfaces	Enables XML-Schema validation for Pro/TK feature XMLs	yes, no	no
enable_shaded_view_in_drawings	Drawing	Yes - shading display is allowed in drawing views. No - shading display will be shown in wireframe.	yes, no	yes
enable_tree_indep	System	Setup whether tree customization will allow separating the tree window from Pro/E window.	yes, no	no
erv_show_external_only	Assembly	Yes - Global Reference Viewer will show objects with external references only;No - all the objects will be visible	yes, no	no
excel_analysis_directory	File Storage & Retrieval	Sets the default directory for the MS Excel files to be used for Excel analysis calculations. Use the full pathname to avoid problems.		
excel_family_table_keep_file	File Storage & Retrieval	Yes - in case of editing family table with excel, if saved excel file exist , will always update it.	yes, no	no
exec_prog_on_assem_retr	File Storage & Retrieval	Execute pro/program for models during assembly retrieval.	yes, no	no
expand_all_with_nested_layers	Without category	Yes - expand all including nested sub layers.	yes, no	yes
explode_animation_max_time	Model Display	Sets the maximum amount of time for an animation between explode states.		1

Name	Category	Description	Values	Default Value
explode_iges_dimension_note	Data Exchange	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
export_3d_force_default_naming	Data Exchange	Specify whether the Pro/ENGINEER model names should be used for STEP export. Yes - Use default naming. No - Add suffixes to model file names.	yes, no	no
export_report_format	System	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
export_to_shipit	Data Exchange	Enable Ship-it interface export.	yes, no	no
extend_cvpath	Data Exchange	Extends CVPATH search path for CADDS5 data exchanging.		
extended_esr_capabilities	Assembly	Extends external simplified representation capabilities.	yes, no	yes
external_analysis_directory	File Storage & Retrieval	Sets the default directory for the files to be used by External Analysis calculations.Use the full pathname to avoid problems.		
fail_ref_copy_when_missing_orig	Assembly	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
fail_unattached_feature	Features	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
fan_significant_locations_only	Electromechanical	Setup whether only significant locations should be flattened on flattening a harness.	yes, no	yes
fan_with_network	Electromechanical	Setup whether the network should be included on flattening a harness.	yes, no	yes
fast_highlight	Model Display	Yes - Use simplified highlighting which redisplays more quickly at the expense of certain details.This option may increase memory usage slightly.	no, yes	yes

Name	Category	Description	Values	Default Value
fasthlr	Model Display	Specifies whether fast hidden line removal is on/off by default.	no, yes	no
feature_create_auto_begin	Features	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
feature_create_auto_ok	Features	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
fem_allow_named_mesh_files	Mechanica	Yes - The user will be prompted to select a file to retrieve the mesh from, or enter a file name to save the mesh to. No - The user will not be prompted to select or enter a mesh file name.	yes, no	no
fem_ansys_annotations	Mechanica	Toggles transfer of loadcase names to ANSYS.	yes, no	no
fem_ansys_grouping	Mechanica	Toggles transfer of groups/layers to ANSYS.	yes, no	no
fem_asp_ratio	Mechanica	The value that FEM created elements of aspect ratio are compared against.		7
fem_default_solver	Mechanica	Specifies default solver.	ansys, nastran	
fem_dist_index	Mechanica	The value that FEM created elements of distortion indices are compared against.		0.4
fem_edge_angle	Mechanica	Sets the minimum acceptable default angle between adjacent element edges. Enter value between 0 and 90.		30
fem_keep_solver_files	Mechanica	Yes - All Solver files will be retained after running the analysis. No - All Solver files will be deleted after running the analysis.	yes, no	no
fem_mesh_operations	Mechanica	Yes - The menu item Operations appears in the FEM MESH menu, providing access to functionalities to manipulate the finite element mesh. No - the menu FEM MESH will not have the item Operations.	yes, no	no
fem_mesh_preserve	Mechanica	Yes - The finite element mesh will be automatically stored in the mesh file upon creation, and retrieved on entering the Structure/Thermal. No - The mesh will not be stored or retrieved automatically.	yes, no	no
fem_mid_ratio	Mechanica	The value that FEM created elements of mid ratios are compared against.		0.1

Name	Category	Description	Values	Default Value
fem_neutral_version	Mechanica	Specifies version number of neutral file.		3
fem_shell_elem_type	Mechanica	Sets the default shell element type in FEM mode.	tri, quad	quad
fem_skew_angle	Mechanica	Sets the maximum acceptable default skew angle value, measured in degrees. Enter a value between 0 and 90.		45
fem_solver_time_limit	Mechanica	For NASTRAN solution, uses this option to interrupt the solver after the specified time limit. The default value is 60 minutes.		60
fem_taper	Mechanica	Sets the minimum acceptable default taper value. Enter a value between 0 and 1.		0.5
fem_warp_angle	Mechanica	Sets the maximum acceptable default warp angle value, measured in degrees. Enter a value between 0 and 90.		10
fem_which_ansys_solver	Mechanica	Specifies the ANSYS equation solver type to be used. Supported solver types: FRONTAL, ITERATIVE (JCG), POWERSOLVER (PCG), ICCG, AUTO, and SPARSE.	frontal, iterative, powersolver, iccg, auto, sparse	
fem_y_dir_compress	Mechanica	Specifies the Y-direction based on compressed and uncompressed geometry. The default value is uncompressed.	yes, no	no
file_dialog_units_class	Without category	Sets the units for manikin attributes displayed in the File Open dialog.	ips, mks	mks
file_open_default_folder	File Storage & Retrieval	Sets default dir for File>Open. Working_directory - Search working directory. In_session - Search objects insession. Pro_library - Use only if Pro/LIBRARY is installed. Workspace - Use only with Pro/INTRALINK.Commonspace - Use only with Pro/INTRALINK.	default, working_directory, in_session, pro_library	
file_open_default_view	File Storage & Retrieval	Specify the initial Views setting as List or Details in the File dialog boxes.	list, details	list
file_open_preview_default	File Storage & Retrieval	Controls whether the preview area is expanded by default when the file open dialog or local file browser is displayed.	expanded, collapsed	collapsed
file_timestamp_format	Without category	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

Name	Category	Description	Values	Default Value
fix_autocad_iges_text_scale	Data Exchange	Fixes scaling problems with AutoCAD releases earlier than Release 10. Yes - Repairs AutoCAD files appropriately.	yes, no	no
fix_catia_iges_sym_note	Data Exchange	Corrects a problem caused by multiple rotations of a note in a CATIA IGES file. Yes - Rectifies any discrepancies.	yes, no	no
fix_imported_set_view_orient	Data Exchange	Sets viewing characteristics for imported CATIA SET files. Yes - Files can have translated views. No - Files cannot have translated views.	yes, no	no
fix_refs_to_intersections	Assembly	Sets remnant removal ability for the ReIntersect command in the ASSY FEAT menu. Yes - You can remove remnants. No - You cannot remove remnants.	yes, no	no
flange_shape_sketches_directory	Sheetmetal	Sets path to directory that contains the Flange shapes.		
flat_shape_sketches_directory	Sheetmetal	Sets path to directory that contains the Flat shapes.		
flip_arrow_scale	Environment	Sets a scale factor for enlarging the size of the flip arrow that appears for feature creation direction (default = 1).		1
floating_layer_tree	Layers	Set to yes if user wants to see a layer tree dialog separated from navigator.	yes, no	no
fonts_size	User Interface	Determines the relative size of the text in menus and dialog boxes. You must restart Pro/ENGINEER for the change to take effect.	small, medium, large	
foot_grid_interval	Features	Modify grid space in foot units for handle movement.		0.083333
force_new_file_options_dialog	File Storage & Retrieval	Force the use of the New File Options dialog.	yes, no	no
force_upd_assem_mp_in_simp_rep	Assembly	Update mass properties parameters even when some of the components of the assembly are not in master rep.	yes, no	no
force_wireframe_in_drawings	Drawing	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
format_setup_file	Drawing	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

Name	Category	Description	Values	Default Value
frames_per_sec	Model Display	Determines whether view is animated.	no, yes	no
frames_per_sec_value	Model Display	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
freeze_failed_assy_comp	Assembly	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. Missing refs using offsets or internal datums do not cause components to freeze.	yes, no	no
freeze_failed_cable_locations	Electromechanical	Suspends the cable location at its last known assembly location.	yes, no	no
frt_enabled	Data Exchange	Loads feature recognition application.	yes, no	no
full_hlr_for_cables	Electromechanical	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
general_undo_stack_limit	System	Setup the default number of entries in the top level Undo/Redo (default = 50).		50
generate_viewable_on_save	File Storage & Retrieval	Controls whether updated ProductView .edz files are automatically exported every time a model is saved.	yes, no	no
geometry_analysis_color_scheme	File Storage & Retrieval	Specifies the location of the geometry analysis color scheme file.		
gerber_360_interpolation	Printing & Plotting	Yes - Arcs and circles included in outputted Gerber plot files will take this format.	no, yes	no
global_appearance_file	Colors	Material Global Appearance file path.		
gpostpp_dir	Manufacturing	Specifies the directory containing the post processor files to be used by Pro/NCPOST (GPOST).		
graphics	System	Sets the optional graphics environment used by certain platforms for running Pro/ENGINEER. Refer to the Installation Guide for a list of defaults and optional graphics.	opengl, x_windows, win32_gdi, d3d	opengl
grid_snap	Sketcher	Yes - Makes pick points snap to a grid. No - Turns the grid snap off so that any location can be picked.	yes, no	no

Name	Category	Description	Values	Default Value
group_repl_with_recycle	Without category	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
group_replace_rename	Without category	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
gtol_dim_placement	Dimensions & Tolerances	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
harn_start_model_dir	Electromechanical	Provides the complete path to the directory containing harness start parts and assemblies		
harn_tang_line_display	Electromechanical	Display the tangency lines between cable segments in thick cable display mode.	yes, no	yes
help_center_path	User Interface	URL of an active Pro/ENGINEER Help Center, if the Help Center is not installed at the default path (along with Pro/E product), of nature http://localhost:/index.jspx. Or, URL of the Pro/ENGINEER Help Center, that is deployed on a web server.		
help_translation	User Interface	Specifies the language in which the short help messages display when running non-English versions of Pro/ENGINEER. Yes - Display in native language. No - Display in English.	yes, no	yes
highlight_erased_dwg_views	Drawing	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
highlight_geometry	Environment	Yes - Items selected in the model tree are highlighted in the geometry area.	yes, no	yes
highlight_layer_items	Layers	Yes - Items selected in the layer tree are highlighted in the geometry area.	yes, no	yes
highlight_new_dims	Drawing	In drawing mode, highlights new dimensions in red until you repaint the screen.	yes, no	no

Name	Category	Description	Values	Default Value
hlr_for_quilts	Drawing	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
hlr_for_xhatches	Electromechanical	Controls whether HLR to be enabled for crosshatches in drawing mode.	no, yes	yes
hole_diameter_override	Features	No- Uses hole-table-driven standard hole diameter value. Yes - Allows specifying the drilled hole value instead of the standard value.	yes, no	no
hole_file_resolution	Features	Specify Hole tool to use external hole parameter (.hol) files.	replace_with_external, use_internal	replace_with_external
hole_parameter_file_path	Features	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.		
iconify_entire_pro	User Interface	Determines how many other Pro/ENGINEER windows/menus are iconified when one window is iconified. Yes - All the windows and menus are iconified. No - Only the selected window is iconified.	no, yes	yes
idd_repair_tangency	Without category	Sets the default state of the "Repair Tangency" checkbox in the Repair and Match tools	yes, no	no
iges_clip_view_ent	Data Exchange	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
iges_clip_view_note	Data Exchange	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
iges_export_dwg_views	Data Exchange	Yes - Exports drawing view information. No - Does not export drawing view information.	yes, no	no
iges_in_assoc_dim_geom_21	Data Exchange	Controls if imported IGES associative dimensions remain associative. Yes - If the drawing setup option associative_dimensioning is Yes, then the Associativity Instance entity is processed during import. No - Dimensions do not retain associativity.	yes, no	no

Name	Category	Description	Values	Default Value
iges_in_dwg_color	Data Exchange	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
iges_in_dwg_line_font	Data Exchange	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
iges_in_group_to_dwg_layer	Data Exchange	Yes - Converts an IGES group to drawing layers. No - The IGES group is not converted.	no, yes	no
iges_note_disp	Data Exchange	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
iges_out_all_srfs_as	Data Exchange	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
iges_out_assembly_default_mode	Data Exchange	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 geominfo of comps and assy feats.	flat, one_level, all_levels, all_parts	flat
iges_out_catia_gdt_width	Data Exchange	Yes - Enables the desired width of a gtol symbol to be exported to CATIA.	yes, no	no
iges_out_catia_notes	Data Exchange	Yes - Breaks a large note (greater than 70 strings) down into smaller notes, each producing an IGES entity.	yes, no	no
iges_out_dwg_color	Data Exchange	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
iges_out_dwg_line_font	Data Exchange	Controls the export of user-defined line font through IGES. No - Exports all geometry as solid font.	yes, no	no

Name	Category	Description	Values	Default Value
iges_out_ent_as_bspline	Data Exchange	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
iges_out_jamais_compliant	Data Exchange	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
iges_out_mil_d_28000	Data Exchange	Yes - Specifies IGES output that uses the MIL-D-28000 entity subset.	yes, no	no
iges_out_spl_crvs_as_126	Data Exchange	Converts all part geometry spline curves to IGES entity 126 (B-spline) when creating an IGES file.	no, yes	yes
iges_out_spl_srfs_as_128	Data Exchange	Converts all part geometry spline surfaces to IGES entity 128 (B-spline) when creating an IGES file.	no, yes	yes
iges_out_start_note	Data Exchange	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
iges_out_symbol_entity	Data Exchange	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
iges_out_trim_curve_deviation	Data Exchange	Sets the maximum value for the distance between an XYZ trimming curve (boundary) and the underlying surface of a trimmed surface		-1
iges_out_trim_xyz	Data Exchange	Controls whether or not XYZ data is output in addition to UV data for trimmed surfaces.	yes, no	yes
iges_out_trm_srfs_as_143	Data Exchange	Yes - Exports all surfaces to IGES entities 141 and 143. This overrides option iges_out_trm_xyz.	no, yes	no
iges_zero_view_disp	Data Exchange	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
ignore_all_ref_scope_settings	Without category	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

Name	Category	Description	Values	Default Value
ignore_export_geom_declaration	Without category	Ignore Export Geometry settings.	yes, no	no
inch_grid_interval	Features	Modify grid space in inch units for handle movement.		0.125
info_output_format	Environment	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
info_output_mode	Environment	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.	choose, screen, file, both	both
initial_bend_y_factor	Sheetmetal	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
initial_drm_warning	User Interface	Warns the user that they are initiating a DRM session.When set to "no", this warning dialog will be skipped.	yes, no	yes
instance_search_exhaustive	File Storage & Retrieval	Specifies extent of instance search. Yes - Include file type when entering instance name in File>Open: type ininput panel (e.g. inst_name.prt) or select in drop-down list. If no .idx files are found, only working dir issearched; each generic is not.	no, yes	no
interface_criteria_mismatch	Without category	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.	warning_only, fail_placement, ignore	warning_only
interface_quality	Data Exchange	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
intf_cadds_import_make_solid	Data Exchange	Allows the user to solidify all closed quilts during import CADDS5 model.	yes, no	no

Name	Category	Description	Values	Default Value
intf_cadds_version	Data Exchange	Allows the user to switch between the default and alternate converter version for interface with CADDS5	14, 15	14
intf_generate_log_file	Data Exchange	Specifies not to generate or generate short or long log files for data exchange import operations.	no, short, long	short
intf_in_dwg_pnt_ent	Data Exchange	Yes - Converts an IGES or DXF point entity to a drawing point.	yes, no	no
intf_in_dwg_view	Data Exchange	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
intf_in_extract_profiles	Data Exchange	Default, no profiles in a data exchange file will be extracted for use.	none, comp, all	none
intf_in_granite_direct_enable	Data Exchange	Allows the user to switch back to the old 'Import Feature' method of reading Desktop .des files and Granite .g files into Pro/E. By default, Pro/E directly opens these models.	yes, no	yes
intf_in_layer_asm_dialog	Data Exchange	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
intf_in_profile_default	Data Exchange	Specifies the name of the existing import profile to use by default.		
intf_out_as_bezier	Data Exchange	Exports all B-splines as Bezier surfaces.	yes, no	no
intf_out_asm_mapped_item	Data Exchange	Export assembly components and instances using MAPPED ITEM entity.	yes, no	no
intf_out_assign_names	Data Exchange	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 Pro/ENGINEER datum points, datum axes, datum curves, surfaces, edges, and quilts.	no_name, user_name, id_name	no_name
intf_out_auto_layer_ids	Data Exchange	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
intf_out_blanked_entities	Data Exchange	Filters export of entities based on their blank status	yes, no	yes
intf_out_cat_start_model	Data Exchange	Specifies Catia start model to be used for CATIA II .model export.		

Name	Category	Description	Values	Default Value
intf_out_ed_recipe_file	Data Exchange	Sets the ProductView recipe file for export to .ed and .edz formats. For use with Windchill 8.0 and earlier servers		
intf_out_layer	Data Exchange	Layer mapping mechanism providing an ability to group entities to layers or blocks.	none, part_layer, block_layer, block_nested_layer, block_view_based_l ayer, block_nested_view_ based_layer	none
intf_out_layer_rename_table	Data Exchange	Enables you to assign interface ids to layers during export		
intf_out_max_bspl_degree	Data Exchange	Controls the maximum degree of exported B-spline surfaces when exporting through IGES		16
intf_out_pvs_recipe_file	Data Exchange	Sets the ProductView recipe file for export to .pvs and .pvz formats. For use with Windchill 9.0 and later servers		
intf_out_text_length	Data Exchange	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
intf_pdf_u3d_create_u3d_files	Without category	YES - When exporting a Pro/E model as PDF U3D file, create a U3D file of the same name as the PDF file.	yes, no	no
intf_profile_dir	Data Exchange	Specifies the directory where import and export profiles are stored. Use the full path to avoid problems.		
intf_pv_recipe_dir	Data Exchange	Sets the directory for user defined ProductView recipe files		
intf_ug_version	Data Exchange	Sets the version of Unigraphics interface.	nx4, nx5	
intf_use_variable_size	Data Exchange	Yes - IGES,DXF,STEP files with variable sized sheet information are imported andput 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 isplaced on a standardsized format.	no, yes	yes

Name	Category	Description	Values	Default Value
intf2d_fit_incompatible_data	Data Exchange	YES - For import and export, fixes compatibility problems between 2-D external formats (such as IGES and DXF) and Pro/ENGINEER.	yes, no	no
intf2d_iges_out_hatch	Data Exchange	YES - Exports draft xhatching as IGES sectioned area entity. NO - exports draft xhatching as separate geometric entities.	yes, no	no
intf2d_in_acad_ignore_3d	Data Exchange	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
intf2d_in_create_multiline_note	Data Exchange	Create a single multi-line note on importing multi-line text when is set to YES.	yes, no	yes
intf2d_in_dxf_mapping_file	Data Exchange	Specifies the mapping file for DXF and DWG import. Absolute or relative path to the file can be used.		
intf2d_in_iges_hatch_bnd_layer	Data Exchange	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
intf2d_in_iges_symbols	Data Exchange	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
intf2d_in_open_log_window	Data Exchange	Yes - Opens Import Log File in a separate window. No - The Log File does not appear in the window.	yes, no	no
intf2d_out_acad_mtext	Data Exchange	Create a single MTEXT entity for multi-line text notes when is set to YES.	yes, no	yes
intf2d_out_acad_ole_as_image	Data Exchange	Controls OLE export to DXF/DWG. Yes - exports OLE objects as DXF/DWG IMAGEs. No - exports OLE objects as separate entities.	yes, no	yes
intf2d_out_acad_text_align	Data Exchange	As_is - Exports notes to DXF/DWG with their original alignment values. Fit - Exports notes with FIT alignment.	as_is, fit	as_is
intf2d_out_acad_unicode	Data Exchange	Yes - Non-ASCII characters are written in UNICODE encoding.	yes, no	no

Name	Category	Description	Values	Default Value
intf2d_out_blanked_layers	Without category	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
intf2d_out_cgm_old_lfont_map	Data Exchange	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
intf2d_out_cgm_ver	Data Exchange	Provides the ability to choose CGM metafile version when exporting from Pro/E drawing.	1, 3	1
intf2d_out_dxf_mapping_file	Data Exchange	Specifies the mapping file for DXF and DWG export. Absolute or relative path to the file can be used.		
intf2d_out_enhanced_ents	Data Exchange	Controls whether the enhanced entities (SPLINE and HATCHING) are converted to the corresponding DXF entities or represented as separate lines and polylines.	spline_and_hatch, spline_only, hatch_only, none	spline_and_hatch
intf2d_out_iges_layers_with_id	Data Exchange	Yes - Exports interface IDs of layers as IGES levels.	yes, no	no
intf2d_out_line_width	Data Exchange	YES - Exports line width information to DXF. NO - all entities are exported with the default width.	yes, no	yes
intf2d_out_open_log_window	Data Exchange	Yes - Opens Export Log File in special window. No - the Log File will not appear in the window.	yes, no	no
intf2d_out_pnt_ent	Data Exchange	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
intf2d_out_stroke_text	Data Exchange	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.	none, special, all	none
intf3d_ideas_import_filter	Data Exchange	Enables UI to selectively import I-DEAS part and assembly models from the I-DEAS .mf1 and .pkg container files.	yes, no	no
intf3d_ideas_install_dir	Data Exchange	Specifies the path for I-DEAS installation. Use the full pathname to avoid problems.		
intf3d_ideas_run_command	Data Exchange	Specifies the I-DEAS run command. Default command is "ideas".		ideas

Name	Category	Description	Values	Default Value
intf3d_in_enable_layer_join	Data Exchange	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
intf3d_in_import_as_facets	Data Exchange	Import faceted data for formats that support both facet and Brep representations.	yes, no, default	default
intf3d_out_annotations	Data Exchange	Export annotations with models.	yes, no, default	yes
intf3d_out_cat2_ident_crv	Data Exchange	Specifies model parameter IDENT_CRV while exporting to Catia.		-1
intf3d_out_cat2_ident_pt	Data Exchange	Specifies model parameter IDENT_PT while exporting to Catia.		-1
intf3d_out_cat2_infinity	Data Exchange	Specifies model parameter INFINITY while exporting to Catia.		-1
intf3d_out_cat2_model_sz	Data Exchange	Specifies model parameter MODEL_SZ while exporting to Catia.		-1
intf3d_out_cat2_sag	Data Exchange	Specifies model parameter SAG while exporting to Catia.		-1
intf3d_out_cat2_step	Data Exchange	Specifies model parameter STEP while exporting to Catia.		-1
intf3d_out_catia_v5_version	Without category	Allows the user to switch between the default and alternate converter version for export to CATIA V5	16, 17, 18	16
intf3d_out_datums_by_default	Data Exchange	Determines whether datum curves are included when exporting IGES files in Batch mode.	yes, no	no
intf3d_out_default_option	Data Exchange	Controls the type of 3-D data output through Pro/BATCH to an IGES or STEP file.	wireframe, surfaces, wireframe_surfaces, solid, shells, none	surfaces
intf3d_out_extend_surface	Data Exchange	Specifies how surfaces are handled for exporting files to other systems	yes, no	yes
intf3d_out_force_surf_normals	Data Exchange	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
intf3d_out_jt_auto_lods	Data Exchange	Generate three default ProE LODs.	no, yes	no
intf3d_out_jt_brep	Data Exchange	Export as faceted data for formats that support both facet and Brep representations.	no, jt_brep, xt_brep	no

Name	Category	Description	Values	Default Value
intf3d_out_jt_config_name	Without category	Name of JT configuration file.		
intf3d_out_jt_config_path	Without category	Path to JT configuration file.		
intf3d_out_jt_structure	Data Exchange	Specifies product structure to file structure mapping defined in JT Open Toolkit.	per_part, fully_shattered, monolithic	per_part
intf3d_out_parameters	Data Exchange	Exports parameters with models.	all, none, designated	all
intf3d_out_prop_chord_heights	Without category	Use maximum chord heights proportional to component sizes.	yes, no	no
intf3d_out_prop_step_sizes	Without category	Use maximum step sizes proportional to component sizes.	yes, no	no
intf3d_out_surface_deviation	Data Exchange	Sets maximum allowable deviation between original and resulting surfaces in the process of converting Pro/ENGINEER surfaces to spline surfaces. By setting this you convert surfaces with a better approximation than by using the current model accuracy.		-1
intf3d_out_unicode_to_neutral	Data Exchange	YES - Unicode strings are exported to the Neutral file, NO - Unicode strings are converted to ascii characters for Neutral file export.	yes, no	yes
intf3d_out_use_num_names	Data Exchange	Use numeric layer names as layer export IDs.	yes, no	yes
intf3d_out_use_step_size	Without category	Use maximum step size(s).	yes, no	no
intf3d_parasolid_export_schema	Data Exchange	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
isogen_attribute_map_file	Piping	Specifies the mapping between isogen parameter and attribute name.		<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_attribute_map.ptd
isogen_endtype_map_file	Piping	Specify name of file mapping Pro/Piping endtypes with ISOGEN endtypes		<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_endtype_map.ptd
isogen_mandatory_attr_file	Piping	Specify file containing mandatory attribute filenames used by ISOGEN.		isogen.fls

Name	Category	Description	Values	Default Value
isogen_nominal_size_map_file	Piping	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches		\<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_nominal_size_map.ptd
isogen_output_files_dir	Piping	Specify directory to save the PCFs created by ISOGEN interface		
isogen_pcf_filename_format	Piping	Specify format to create filenames of PCF files being created		mnemonic,-,specification,-,number
isogen_symbol_map_file	Piping	Specify name of file for mapping Pro/Piping selection names with ISOGEN symbol keys (SKEY)s		\<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_symbol_map.ptd
jlink_java_command	Without category	Specifies a command to start JRE for J-Link. May contain arguments and environment variables. Overrides the default, see also jlink_java2.		
jlink_java2	Application Programming Interfaces	Option to force use of Java-2 JRE command line ("java" instead of "jre")	off, on	on
kbd_cmd_abbreviation	User Interface	On - Enables use of abbreviations when entering menu commands from the keyboard.	on, off	off
kbd_selection	User Interface	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.	no, yes	no
keep_info_datums	Features	Sets the initial state of how the system treats datums and features created at run time.	yes, no	yes
keep_model_in_open_window	Without category	Controls whether a model is displayed in its window whenthe accessory window is open.	no, yes	no
keep_style_items_hidden_on_exit	Without category	Keep style entities hidden after exiting of the Style feature definition.	yes, no	no
lang_propagate	File Storage & Retrieval	Specifies language storage options. The option provides flexibility for objects that use a different language than the current	yes, no	yes
last_session_directory_path	Without category	Sets the directory that holds the temp files for the session recording mechanism.		/work_dir/.wf/.settings

Name	Category	Description	Values	Default Value
last_session_retrieval	Without category	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
layer_item_highlight_limit	Layers	Set the upper limit of items in a layer for highlighting.If items in a layer exceeds this number, highlighting will not occur.		500
layer_rule_retrieve_objects	Without category	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.	promt, ok, cancel	promt
let_proe_rename_pdm_objects	Data Management	Determines whether an object fetched from a PDM database can be renamed in a Pro/Engineer session.	no, yes	no
linear_tol	Dimensions & Tolerances	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
linear_tol_0.0	Dimensions & Tolerances	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
linear_tol_0.00	Dimensions & Tolerances	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
linear_tol_0.000	Dimensions & Tolerances	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
linear_tol_0.0000	Dimensions & Tolerances	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

Name	Category	Description	Values	Default Value
linear_tol_0.00000	Dimensions & Tolerances	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
linear_tol_0.000000	Dimensions & Tolerances	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
lods_enabled	Model Display	Determines whether or not the system uses Level of Detail in shaded models during dynamic orientation (panning, zooming and spinning).	no, yes	no
lods_value	Model Display	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
maintain_limit_tol_nominal	Dimensions & Tolerances	Maintains the nominal value of a dimension regardless of the changes that you make to the tolerance values.	yes, no	no
make_aux_view_notes	Drawing	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
make_parameters_from_fmt_tables	Drawing	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
make_proj_view_notes	Drawing	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
manikin_drag_auto_locks	Without category	Manikin drag auto locks	yes, no	yes
manikin_flexibility	Without category	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.	full, none, default	default
manikin_replace_keep_posture	Without category	Controls whether a manikin's existing posture is applied to a replacement manikin.	yes, no	no

Name	Category	Description	Values	Default Value
mapkey	Miscellaneous	Stores a keyboard macro. Use Tools>Mapkeys to create, edit and delete mapkeys.		
mapkeys_execution	Miscellaneous	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
mark_approximate_dims	Dimensions & Tolerances	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
mass_property_calculate	Environment	Calculate mass properties upon regeneration or by user request.	by_request, automatic	by_request
mat_assign_appearance	Colors	Control whether default appearance in material definition is automatically assigned to a part.	yes, no	yes
material_upd_smt_bend_table	Sheetmetal	Defines whether assignment of material to the model via Pro/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
mathcad_analysis_directory	File Storage & Retrieval	Sets the default directory for the Mathcad files to be used for Mathcad analysis calculations. Use the full pathname to avoid problems.		
mathcad_automapping	Environment	Toggles Mathcad analysis automapping of variables	yes, no	no
max_animation_time	Model Display	When the orientation, or zoom state, of the Pro/ENGINEER window changes, an animation sequence of a predefined duration takes place. This option controls the maximum time the system spends animating.		1
max_image_dimension	Model Display	Set the upper bound dimension on images loaded into Pro/ENGINEER. Images larger than this limit will be rescaled to this upper bound during loading.	4096, 1024, 2048, 8192, 16384	4096
mdl_tree_cfg_file	User Interface	Specifies the Model Tree configuration file to be loaded when Pro/ENGINEER is started.		
mdo_cl_executable	Mechanism	Path for Custom Load executable.		
mdo_explicit_integration	Mechanism	Enable explicit integration for dynamic analyses.	yes, no	no

Name	Category	Description	Values	Default Value
mdo_spring_wire_radius	Mechanism	Spring wire radius for dynamics spring icons.		0.15
mdx_belt_icon_width	Mechanism	Width of belt transmission icons.		5
mdx_collision_num_triangles	Environment	The number of mesh triangles which are used to approximate model in collision detection.		1000
measure_dec_places	Without category	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
measure_sig_figures	Environment	Sets the number of significant figures displayed as results when using the Measure dialog box from the Info menu.		6
medusa_2d_config_file	Data Exchange	Specifies the Medusa 2D Interface config. file. Absolute or relative path to the file can be used.		
mentor_ver_2_0	Data Exchange	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
menu_font	User Interface	Specify font used in the Pro/ENGINEER 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.		
menu_manager_position	User Interface	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.	outside, inside, adaptive	adaptive
menu_mapper_location	User Interface	Specify your own URL for the Menu Mapper button in the help menu.		
menu_show_instances	File Storage & Retrieval	Determines whether instance names listed in instance index files appear in file lists.	yes, no	yes
menu_translation	User Interface	Specifies the language in which the menus display when running non-English version of Pro/ENGINEER.	yes, no, both	yes

Name	Category	Description	Values	Default Value
merge_smt_srfs_without_seam	Sheetmetal	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
merge_type	Features	Sets the default merge type surface merge operations.	join, intersect	intersect
mesh_spline_surf	Model Display	Turns the blue mesh surface lines on or off.	no, yes	no
meter_grid_interval	Features	Modify grid space in meter units for handle movement.		0.001
mfg_auto_ref_mfg_template	Without category	Enable/Disable auto referencing for Manufacturing template	yes, no	no
mfg_auto_ref_prt_as_chk_srf	Manufacturing	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
mfg_auto_retract_clear_dist	Without category	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.		-0.001
mfg_custom_tool_param_file	Manufacturing	Defines the file that contains the user-defined tool parameters' list.		
mfg_customize_dialog	Manufacturing	Yes - Uses the new customize dialog box for customizing NC sequences. No - Uses the Release 19.0 menus.	yes, no	yes
mfg_cutline_save_mesh	Manufacturing	Yes - Save / Retrieve Mesh for subsequent cutline recalculations, No - New Mesh will be computed for each cutline recalculations.	yes, no	yes
mfg_ijk_num_digits	Manufacturing	Sets the number of digits for i, j, k data points in a CL data file.		10
mfg_info_location	Manufacturing	top left /bottom right -Position of the mfg. Info dialog box.	top_left, bottom_right	top_left
mfg_modal_command_file	Manufacturing	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.		
mfg_param_auto_copy_from_tool	Manufacturing	Do not copy any parameter of the tool to the sequence parameter tree.	none, all, misc, cutting	none

Name	Category	Description	Values	Default Value
mfg_process_print_dir	Without category	Specifies the directory where Mfg Process html page get stored. Use the full pathname to avoid problems.		
mfg_process_table_setup_dir	Without category	Specifies the directory where Mfg Process Table settings get stored. Use the full pathname to avoid problems.		
mfg_process_template_dir	Without category	Specifies the directory where Mfg Process Templates get stored. Use the full pathname to avoid problems.		
mfg_setup_dir	Without category	Specifies the directory which contains all manufacturing related setup files for the current session.		
mfg_start_model_dir	File Storage & Retrieval	Provides the complete path to the directory containing mfg start assemblies.		
mfg_tap_mat_rem_coef	Without category	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
mfg_template_dir	Manufacturing	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.		
mfg_tool_dialog_setup_file	Without category	Specifies the XML based file to setup Tool Dialog.		
mfg_udf_info_setup_file	Without category	Specify name and path of the file containing udf and mfg criteria values .		
mfg_workpiece_transparency	Manufacturing	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.		0.5
mfg_xyz_num_digits	Manufacturing	Sets the number of digits for x, y, z data points in a CL data file.		10
millimeter_grid_interval	Features	Modify grid space in millimeter units for handle movement.		1

Name	Category	Description	Values	Default Value
min_animation_steps	Model Display	When the orientation, or zoom state, of the Pro/ENGINEER window changes, an animation sequence of predefined duration takes place. This option controls the minimum number of frames used in the animation sequence.		6
minimum_angle_dimension	Dimensions & Tolerances	Controls the minimum angle value under which Pro/ENGINEER assumes colinearity of the selected items when creating a driven dimension.		1
model_allow_ref_scope_change	Without category	Yes - Users can change the scope setting of components.	yes, no	yes
model_grid_balloon_display	Environment	Controls the display of grid balloons.	yes, no	yes
model_grid_neg_prefix	Environment	Enables you to specify the prefix to use for negative numbers in the balloon capture of the grid.		-
model_grid_num_dig	Environment	Controls the number of decimal places displayed in the balloon capture. 0 - Only integers display.		
model_grid_spacing	Environment	Sets the default model grid spacing.		1.5
model_notes_as_labels	Environment	Gives the option to display notes as either full text or as label. Can also be toggled with environment option "Display Notes as Names."	no, yes	no
model_rename_template	File Storage & Retrieval	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 [*a1*=*b1*] changes every occurrence of a1 to b1 (e.g. TOP_A1_BRACKET copies to TOP_B1_BRACKET).		
model_tree_font	Without category	Specify font used in the Pro/ENGINEER 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.		
model_tree_start	Environment	Toggles display of the Model Tree. Yes - Displays the Model Tree window along with its model. No - Does not display the Model Tree window.	no, yes	yes
modelcheck_dir	Without category	Specifies path to the ModelCHECK configuration files.		
modelcheck_enabled	Environment	Enable ModelCHECK	yes, no	yes
modeltree_show_gen_of_instance	User Interface	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

Name	Category	Description	Values	Default Value
modified_objs_in_new_dir	File Storage & Retrieval	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
mold_layout_origin_name	Casting & Mold Design	Sets a specified coordinate system as the default for the cavity layout origin.		
mold_vol_surf_no_auto_rollback	Casting & Mold Design	Do not perform automatic rollback while modifying mold volumes or parting surfaces.	yes, no	no
motion_envlp_alert	Mechanism	Displays the Motion Envelope alert the first time the quality level is increased.	yes, no	yes
move_mechanism_in_nongeom_reps	Mechanism	Allow mechanism operations in non geometrical representations.	no, yes	yes
mp_calc_level	Without category	Specifies the level of mass properties calculation.	assembly_only, all_models	assembly_only
mp_dens_ft_column_mtrl_assigned	Assembly	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.	no, yes	no
msg_translation	User Interface	Specifies language in which Messages are displayed when running non-English version of Pro/ENGINEER.	yes, no	yes
mtrl_convert_units	Assembly	If set to "Yes" Pro/E will automatically convert units in materials from a library to the model system of units.	yes, no	yes
multiple_skeletons_allowed	Without category	Allows/restricts multiple skeleton.	yes, no	no
native_kbd_macros	User Interface	Specifies support of keyboard macros written in native language (for example, German).	no, yes	no
nc_autodoc_param_dir	Manufacturing	Location of the setup file that describes which process parameters are included in the parameter table.		
nc_autodoc_report_dir	Manufacturing	Location for storing the report and images generated by the shop floor documentation tool.		
nc_autodoc_setup_dir	Manufacturing	Location for documentation setup files that describe which process parameters are included in the parameter table.		
nc_autodoc_template_dir	Manufacturing	Location of the style sheet and default images used for shop floor documentation.		

Name	Category	Description	Values	Default Value
nc_jobman_visible	Without category	Yes - make "Submit Path" and "Abort Path" visible. No - make "Submit Path" and "Abort Path"invisible.	yes, no	no
nccheck_type	Manufacturing	Vericut will be used for NC material removal simulation.	vericut, nccheck	vericut
ncl_file_extension	Manufacturing	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
ncmdl_bar_stock_part_path	Manufacturing	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<proe_WF_5.0_loadpoint>/text/ncmdl_data/ncmdl_bar.prt
ncmdl_billet_stock_part_path	Manufacturing	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<proe_WF_5.0_loadpoint>/text/ncmdl_data/ncmdl_billet.prt
ncpost_type	Manufacturing	Set to GPOST to access the default post processor supplied with Pro/NCPOST.	gpost, ncpost	
ncseq_outbnd_curve	Without category	Yes - Create out bound curve.	no	no
ncseq_outbound_curve	Manufacturing	Yes - Create out bound curve.	yes	
new_parameter_ui	User Interface	Enable the new parameter editor and user interface.	yes, no	yes
new_ref_part_name_disabled	Without category		yes, no	yes
new_relation_ui	User Interface	Enable the new relations editor and user interface.	yes, no	yes
new_turn_scan_type	Manufacturing	Allows to use New turn scan types for Turning NC sequences.	yes, no	yes
new_wf5_simp_rep_ui	Without category	Yes - Use Component Chooser base Simp Rep UI, No - Use old Simp Rep definition UI.	yes, no	yes
number_user_colors	Colors	Specifies the maximum number of different wireframe colors that can display in the graphics area(s) at any given moment.		-1
offset_line_def_color	Assembly Process	Specifies the color that offset lines are displayed in drawings.		0
offset_line_def_font	Assembly Process	Specifies the font in which offset lines are displayed in drawings.		
old_style_set_datum	Without category	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

Name	Category	Description	Values	Default Value
online_resources_location	User Interface	Specify your own URL for the Online Resources button in the help menu.		
open_draw_simp_rep_by_default	File Storage & Retrieval	Set to "Yes" to always invoke the open rep dialog when opening a drawing.	yes, no	no
open_simplified_rep_by_default	File Storage & Retrieval	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
orientation	Model Display	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
orientation_style	Without category	Set spin orientation style to dynamic or anchored.	anchored, dynamic	dynamic
orthogonal_snap	Electromechanical	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
output_mfg_xml	Without category	Allows export/import mfg model to/from xml file. Yes- make Export/Import options visible. No - make them invisible.	yes, no	no
overlays_enabled	System	Yes - Enables Pro/ENGINEER screens and menus to be placed on different overlay layers of the hardware graphics card. This frees memory for Pro/ENGINEER use.	yes, no	no
override_store_back	File Storage & Retrieval	Forces the storage of objects retrieved from other directories to the current working directory.	yes, no	no
package_constraints	Assembly	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
package_ref_alert	Assembly	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
param_dec_places	User Interface	Sets the number of decimal places (0-14) to display in parameters.		6

Name	Category	Description	Values	Default Value
parameter_dialog_setup_file	User Interface	Setup for the parameter dialog configuration file.		
parenthesize_ref_dim	Dimensions & Tolerances	Encloses reference dimensions in parentheses. No - Follows reference dimensions with the text "REF".	yes, no	no
pattern_exact_incremental_dim	Without category	When pattern incremental dimensions are calculated from expressions, round them to the decimal places set by config option default_dec_places.	no, yes	no
pattern_relations_copy	User Interface	Determines whether relations of pattern leader are inherited by pattern members.	yes, no	yes
pdf_linecap	Data Exchange	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
pdf_linejoin	Data Exchange	0 - Miter join: outer edges of the strokes for the two segments are extended until they meet at an angle.	miter, round, bevel	miter
pdf_use_pentable	Data Exchange	No - PDF export does not use any defined pentable changes as would be applied to plotting the drawing.	no, yes	no
pdm_rev	Data Management	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
pdm_server_quickview_rows_num	Without category	Sets the default number of rows for the pdm quick view list.		7
pen_slew	Printing & Plotting	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
pen_slew_xy	Printing & Plotting	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
pen_table_file	Printing & Plotting	Specifies a default pen mapping table which supersedes other pen mapping information.		table.pnt
pen1_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		4

Name	Category	Description	Values	Default Value
pen2_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		1
pen3_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		2
pen4_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		3
pen5_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		2
pen6_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		3
pen7_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		1
pen8_line_weight	Printing & Plotting	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).		4
photorender_capability_warnings	Photorender	Allows the user to disable warnings about PhotoLux only capabilities.	yes, no	yes
photorender_default_height	Photorender	Sets the default height (in pixels) for the custom size rendered output		450
photorender_default_width	Photorender	Sets the default width (in pixels) for the custom size rendered output		600
photorender_memory_usage	Photorender	Sets the memory limit that Pro/PHOTORENDER allows for model processing. Do not use commas in the memory value, as Pro/ENGINEER interprets these as decimals.		256
photorender_preview_scale	Photorender	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

Name	Category	Description	Values	Default Value
pick_aperture_radius	System	Specifies the size of the area about the mouse when making selections. Units are 1/1000 of screen size.		7
pick_chain_tangent_only	Drawing	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
pipe_3d_bend_theor_int_pts	Without category	Yes - Displays theoretical intersections and bends in three-dimensional models.	no, yes	no
pipe_bend_locations_csys_units	Piping	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
pipe_pre_22_bendinfo_conv	Piping	Convert bend report information generated prior to Pro/ENGINEER 2000i-2 to current saved analysis format.	yes, no	yes
pipe_solid_centerline	Piping	Yes - Display Pro/PIPING centerlines. No - Do not display Pro/PIPING centerlines.	yes, no	yes
pipe_solid_label_format	Piping	Specify the pipe solid label format that will be used to automatically generate pipeline labels.		size,-,specification,-,mnemonic,-,number,-,insulation
pipe_update_pre_20_int_pts	Piping	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
pipeline_assembly_library_dir	Piping	Sets the default pipeline assembly library directory.		<proe_WF_5.0_loadpoint>/text/piping_data/pipelinelib/
pipeline_assembly_name_format	Piping	Sets the default pipeline assembly name format.		mnemonic,-,number
pipeline_label_format	Piping	Specify the pipeline label format that will be used to automatically generate pipeline labels.		size,-,specification,-,mnemonic,-,number,-,insulation
pipeline_solid_start_part_name	Piping	Sets the default pipeline solid start part name.		pipeline_solid_start.prt
pipeline_spool_label_format	Piping	Specify the pipeline spool label format that will be used to generate the spool labels during pipeline spool generation.		mnemonic-number-prefix[0]spoolnum[01]suffix[]

Name	Category	Description	Values	Default Value
pipeline_start_assembly_name	Piping	Sets the default pipeline start assembly name.		pipeline_start.asm
piping_appearance_map_file	Piping	Sets the default piping appearance map filename.		\<proe_WF_5.0_loadpoint\>/text/piping_data/master_catalog/piping_appearance.map
piping_bolt_nut_select_file	Piping	Piping bolts and nuts selection file.		\<proe_WF_5.0_loadpoint\>/text/piping_data/master_catalog/piping_bolt_nut_select.ptd
piping_design_method	Piping	Sets the current Piping design method for pipeline modeling.	non_spec_driven, spec_driven, user_driven	non_spec_driven
piping_enable_designate_report	Piping	Enable designatable reports for spec-driven piping.	yes, no	yes
piping_end_compatibility_file	Piping	Sets the default piping end compatibility filename.		\<proe_WF_5.0_loadpoint\>/text/piping_data/master_catalog/piping_end_compatibility.ptd
piping_extend_dim_scheme	Piping	Specify a dimension scheme for piping extend segment.	cartesian, cylindrical, spherical	cartesian
piping_extend_show_csys	Piping	Control whether small preview CSYS is shown at end of pipe when defining extend features in cartesian / spherical / cylindrical schemes.	yes, no	no
piping_fitt_angle_tolerance	Piping	Sets the default piping fitting angle tolerance value.		1.5
piping_fitt_category_map_file	Piping	Sets the default piping fitting category map filename.		\<proe_WF_5.0_loadpoint\>/text/piping_data/sample_project/piping_fitt_category_map.ptd
piping_fitt_library_dir	Piping	Sets the default piping fitting library directory.		\<proe_WF_5.0_loadpoint\>/text/piping_data/fittinglib/
piping_fluid_parameter	Piping	Enables/disables the Parameters tab in Create Pipeline Dialog	no, yes	no

Name	Category	Description	Values	Default Value
piping_insulation_dir_file	Piping	Sets the default piping insulation directory filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/sample_project/piping_insulation_dir.ptd
piping_joint_fitting_clearance	Piping	Sets the piping joint minimum fitting clearance value.		1
piping_joint_fitting_offset	Piping	Sets the piping joint minimum fitting offset value.		1
piping_manufacture_dir_file	Piping	Sets the default piping manufacture directory filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_manufacture_dir.ptd
piping_material_file	Piping	Sets the default piping material filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_material.ptd
piping_mcat_dir	Piping	Sets the default piping master catalog directory.		\<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/
piping_mcat_dir_file	Piping	Sets the default piping master catalog directory filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_mcat_dir.ptd
piping_min_bolting_clearance	Piping	Sets the default piping minimum bolting clearance value.		1
piping_project_data_dir	Piping	Sets the default Project data directory.		\<proe_WF_5.0_loadpoint>/text/piping_data/sample_project/
piping_schematic_driven	Piping	Set the Schematic Driven Pipeline Modeling ON/OFF	no, yes	no
piping_schematic_xml_dir	Piping	Specify directory with full path containing schematic information XML file(s).		/work_dir/config_pro/
piping_spec_dir_file	Piping	Sets the default piping specification directory filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/sample_project/piping_spec_dir.ptd
piping_system_tree_format	Piping	Sets the default piping model tree format.		mnemonic
piping_thumb_wheel_increment	Piping	Sets the default piping thumb wheel increment value.		1

Name	Category	Description	Values	Default Value
piping_thumb_wheel_ratio_inc	Piping	Sets the default piping thumb wheel ratio increment value.		0.01
piping_wildfire_convert	Piping	Always ask before converting old piping assemblies to update the network structure.	manual, never, automatic	manual
placement_rep_by_default	File Storage & Retrieval	Retrieves the assembled component in a specified representation. None - retrieves the representation specified in configuration option open_simplified_rep_by_default.		none
plot_file_dir	Printing & Plotting	Specifies the directory where plot files should be written. Use the full pathname to avoid problems. For example /home/users/plotfiles.		
plot_linestyle_scale	Printing & Plotting	Specifies the scaling factor for the DOTFONT line style in a plot.		1
plot_names	Printing & Plotting	Yes - Plot files are given descriptive extensions when created. No - All plot files are simply given the extension .plt.	yes, no	
plot_names_with_sheet	Without category	Yes - Plot file names are appended with _(sheet number) before file extension.	yes, no	no
plot_proceed_dialog	Printing & Plotting	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
plot_to_scale_full_window	Printing & Plotting	Provides the ability to exclude empty window space when plotting the model to scale.	yes, no	
plotter	Printing & Plotting	Establishes the default plotter for making plot files.		
plotter_command	Printing & Plotting	Sets the command that you use to start a plot on your system; for Windows NT or Windows 95, windows_print_manager option configures Pro/ENGINEER to plot to a recognized device in the Windows NT or Windows 95 print manager.	windows_print_manager	
plotter_handshake	Printing & Plotting	Specifies the type of plotter handshake initialization sequence generated in the plotter files.	hardware, software	
popup_refpairintch_window	Without category	Bring forward the window being used with the reference pairing table.	yes, no	no

Name	Category	Description	Values	Default Value
popuphelp_font	User Interface	Specify font used in the Pro/ENGINEER 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.		
postscript_compression	Printing & Plotting	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
preferred_publish_deliverable	Drawing	Set the default preferred deliverable for the Publish Context in Drawings.	print, iges, set, dxf, step, cgm, pdf, dwg, tiff, medusa, stheno	print
preferred_save_as_type	File Storage & Retrieval	Allows customizing the order of the Type selections found under File ->Save A Copy.		
prehighlight	Environment	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	
prehighlight_tree	Environment	Yes ? Selectable items in the model tree and layer tree are highlighted prior to being selected.	yes, no	no
preserve_comp_color_in_preview	Assembly	Yes - During component placement, component being assembled will retain its normal colors.No - During component placement, component being assembled is shown in preview color.	no, yes	no
prev_proc_comp_def_color	Assembly Process	Specify the default color for components added to a process assembly in a previous step.		0
prev_proc_comp_def_font	Assembly Process	Specify the default font for previously added components in a process assembly.		
prime_analysis_directory	Without category	Sets the default directory for the Prime files to be used for Prime analysis calculations. Use the full pathname to avoid problems.		
prime_automapping	Without category	Toggles Prime analysis automapping of variables	yes, no	no
pro_ansys_path	Mechanica	Specifies path to ANSYS executable.		

Name	Category	Description	Values	Default Value
pro_catalog_dir	Casting & Mold Design	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.		
pro_cav_lay_rule_dir	Casting & Mold Design	Sets the default directory for cavity layout rules. Use the full path name to avoid problems.		
pro_cbltrm_dir	Electromechanical	Sets the directory from which terminators are retrieved. Use the full pathname to avoid problems.		
pro_colormap_path	Colors	Specifies the directory path for a color map (.map) file to be loaded from disk. Use the full path name to avoid problems.		
pro_crosshatch_dir	File Storage & Retrieval	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.		
pro_dtl_setup_dir	Drawing	Sets the directory for the drawing setup files. If this option is not set, the system uses the default setup directory.		
pro_editor_command	User Interface	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.		
pro_font_dir	File Storage & Retrieval	Sets the default font directory (Pro/ENGINEER will load all fonts in this directory into the drawing regardless of the contents of the system and working directories).		<proe_WF_5.0_loadpoint>/text/fonts/
pro_format_dir	Drawing	Sets the default directory for the drawing format library. Use the full pathname to avoid problems.		
pro_gplug_dir	Data Exchange	Specifies the directory in which Granite application plug-ins are located. Use the full pathname to avoid problems.		<proe_WF_5.0_loadpoint>/sun_solaris_x64/gplugs/
pro_group_dir	Features	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.		

Name	Category	Description	Values	Default Value
pro_insulation_dir	Piping	Specifies the directory to search for insulation files used in Pro/PIPING. Use full path name to avoid problems.		
pro_library_dir	File Storage & Retrieval	Sets the default directory for the Pro/ENGINEER libraries (or any libraries you have built containing the proper index menu files). Use the full pathname to avoid problems.		
pro_material_dir	File Storage & Retrieval	Sets the default directory for the part material library. Use the full pathname to avoid problems.		
pro_mdb_dir	Manufacturing	Sets the default directory for the Machinability Database (MDB) files. Use the full pathname to avoid problems.		
pro_mf_cl_dir	Manufacturing	Sets the default directory for the manufacturing CL data files.Use the full pathname to avoid problems.		
pro_mf_clamp_dir	Manufacturing	Specifies the directory where the manufacturing clamps can be found.		
pro_mf_param_dir	Manufacturing	Sets the default directory for the manufacturing parameter files. Use the full pathname to avoid problems.		
pro_mf_tape_dir	Manufacturing	Sets the default directory for the manufacturing TAPE data files.Use the full pathname to avoid problems.		
pro_mf_tprm_dir	Manufacturing	Sets the default directory for the manufacturing tool files. Use the full pathname to avoid problems.		
pro_mf_workcell_dir	Manufacturing	Sets the default directory for the manufacturing workcell files. Use the full pathname to avoid problems.		
pro_mfg_cmdsyn_dir	Manufacturing	Sets the default directory for the syntax menu files in Pro/MFG. Use the full pathname to avoid problems.		
pro_nastran_path	Mechanica	Specifies the path to the MSC/NASTRAN executable (nastran).		
pro_note_dir	Drawing	Specifies the directory from which to retrieve notes entered from a file. Use the full path name to avoid problems.		/work_dir/config_pro/
pro_palette_dir	Drawing	Sets the default directory for drawing symbol palette files.		<proe_WF_5.0_loadpoint>/symbols/palette/

Name	Category	Description	Values	Default Value
pro_pip_fitt_dir	Piping	Specifies the directory to search for fittings used in Pro/PIPING.		
pro_pip_lnstk_dir	Piping	Specifies the directory to search for linestock files used in Pro/PIPING. Use full path name to avoid problems.		
pro_plot_config_dir	Printing & Plotting	Sets the directory of your user-defined plotter configuration file. Use the full pathname to avoid problems.		
pro_sheet_met_dir	Sheetmetal	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.		
pro_smt_params_dir	Sheetmetal	Specifies the directory to save/retrieve Sheet metal parameters files.Use full path name to avoid problems.		
pro_solver_name	Mechanica	Specifies the name of the user-defined solver to be included in the SOLVE menu. See also pro_solver_path.		
pro_solver_path	Mechanica	Specifies the path to the user-defined solver whose name is defined in the SOLVE menu. See also pro_solver_name		
pro_spool_dir	Electromechanical	Sets the directory from which spools are retrieved. Use the full pathname to avoid problems.		/work_dir/config_pro/
pro_stheno_command	Data Exchange	Specifies the command for Stheno launch.		
pro_surface_finish_dir	File Storage & Retrieval	Sets the default directory for your user-defined surface finish symbols. Use the full pathname to avoid problems.		<proe_WF_5.0_loadpoint>/symbols/surffins/
pro_symbol_dir	Drawing	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.		/work_dir/config_pro/
pro_texture_library	Photorender	Allows the user to specify a different graphics library.		
pro_unit_length	Environment	Sets the default units for new objects.	unit_inch, unit_foot, unit_mm, unit_cm, unit_m	unit_inch

Name	Category	Description	Values	Default Value
pro_unit_mass	Environment	Sets the default units for mass for new objects.	unit_ounce, unit_pound, unit_ton, unit_gram, unit_kilogram, unit_tonne	unit_pound
pro_unit_sys	Environment	Specifies the default system of units for new models.	mks, cgs, mmns, fps, ips, proe_def, mmks	
pro_weld_params_dir	Weld	Specifies the directory to search when a weld parameter file is needed.		
prodevdat	Application Programming Interfaces	Specify the name of the Pro/Develop registry file.		
proe_memory_buffer_size	Environment	Specify the size in megabytes of the memory buffer to be reserved by Pro/ENGINEER for system out of memory condition. You must restart Pro/ENGINEER for the change to take effect.		50
profile_output_utils	Manufacturing	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
prompt_on_erase_not_disp	File Storage & Retrieval	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
prompt_on_exit	User Interface	Determines if you are prompted to save objects when exiting a Pro/ENGINEER session.	yes, no	no
proncpost_dbf	Manufacturing	Enter directory path to NCPost database file.		
propagate_change_to_parents	File Storage & Retrieval	When saving objects with option save_objects set to Changed or Changed_and_specified, Pro/ENGINEER 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
propagate_inactive_annotation	User Interface	Yes - causes the propagation of inactive annotation.	no, yes	yes

Name	Category	Description	Values	Default Value
prorembatch_queue_manager	Manufacturing	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.		
protable_in_background	System	Yes - Starts Pro/TABLE editor in the background. Reduces time between pressing a menu button that uses Pro/TABLE and entering the data.	no, yes	yes
protkdat	Application Programming Interfaces	Specify the Pro/TOOLKIT registry file.		
provide_pick_message_always	User Interface	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
ps_output_with_pagesize	Printing & Plotting	Specifies whether or not to include pagesize information when generating postscript files from Pro/ENGINEER. No - Pro/ENGINEER will omit Adobe level 2 pagesize information from the header of the postscript file.	yes, no	yes
ptc_manikin_library_path	Without category	Sets the default directory for the Pro/ENGINEER manikin library (or custom manikin libraries you built containing the proper index menu files). Use the full path name to avoid problems.		<proe_WF_5.0_loadpoint>/manikin_lite/
ptc_manikin_posture_path	Without category	Sets the default directory for the Pro/ENGINEER manikin postures (or custom posture libraries you built). Use the full path name to avoid problems.		
ptncpost_dbf	Manufacturing	Enter directory path to NCPost database file.		
punch_axis_points	Sheetmetal	Yes - Enables the creation of punch axis points in Sheetmetal cuts and punches.	yes, no	no
punch_models_directory	Sheetmetal	Sets path to directory that contains the punch models.		
put_iges_drawing_entity	Data Exchange	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
qty_balloon_change_warning	Without category	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

Name	Category	Description	Values	Default Value
quick_print_drawing_template	Without category	Specify the path and name of a drawing template to be used by the File > Quick Print command.		
quick_print_plotter_config_file	Without category	Specify the path and name of the default plotter config file to be used by the File > Quick Print command.		
radial_hole_linear_dim	Without category	YES - Option for creation of radial hole with a linear dimension will be available.	yes, no	no
raster_plot_dpi	Printing & Plotting	Determines the resolution for plotter outputs variables are in dots per inch.	100, 200, 300, 400	100
ratio_grid_interval	Features	Modify grid space in percentage units for handle movement.		0.05
read_famtab_file_on_retrieve	File Storage & Retrieval	No - Ignores filename.ptd. Yes - Creates and saves filename.ptd and uses that file when retrieving the generic.	no, yes	no
read_parameters_of_excluded	Data Management	Allow Pro/ENGINEER designated parameters on excluded objects in Simplified Representation to be visible in Model Tree columns.	yes, no	no
real_time_rendering_display	Model Display	Disable realtime rendering.	disable, transparent_floor, room_display	disable
recompute_iges_dim_value	Data Exchange	Indicates whether to recompute associative dimensions upon import of an IGES file.	yes, no	no
ref_scope_copy_color	Without category	Set the color of backed up references according to RGB values. Enter RGB values between 0 and 100 separated by space.		55
ref_scope_copy_color_change	Without category	Enable changing colors of backuped references during selection.	no, yes	yes
ref_scope_no_pick_to_copy	Without category	Disable selecting references backuped by references control.	no, yes	no
ref_scope_no_pick_to_prohibit	Without category	Disable selecting references prohibited by references control.	no, yes	yes
ref_scope_prohibit_color	Without category	Set the color of prohibited references according to RGB values. Enter RGB values between 0 and 100 separated by space.		45
ref_scope_prohibit_color_change	Without category	Enable changing colors of prohibited references during selection.	no, yes	yes

Name	Category	Description	Values	Default Value
regen_backup_directory	Environment	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.		
regen_backup_using_disk	Environment	Specifies whether or not the system backs up the current model to disk before each regeneration.	yes, no	no
regen_failure_handling	Without category	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
regen_int_mdls_on_retr	Without category	Yes to regen models intersected with assem feats during assembly retrieval.	yes, no	no
regen_layout_w_assem	Assembly	Specifies whether or not the layout will be automatically regenerated upon assembly regeneration.	yes, no	yes
regen_simp_retrieve	File Storage & Retrieval	Regenerate the assembly when retrieving an assembly simplified representation. Yes - Regenerates the assembly on retreival.No - The assembly is not regenerated on retrieval.	yes, no	no
regenerate_read_only_objects	Data Management	Determines whether an object retrieved from PDM database with read only status can be regenerated in a Pro/ENGINEER session.	yes, no	yes
relation_file_editor	User Interface	Sets the editor used to edit relations.	protab, editor	editor
relation_text_trail_output	Miscellaneous	No - Disable trail output from Text layout in Relation Parameter UI if it hasn't been changed.	yes, no	yes
relation_tool_mapkey_behavior	Miscellaneous	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
relations_num_const_units	Without category	Specifies whether diagnostic should be issued when an unitless numeric value causes relation expression evaluation trouble.	yes, no	no

Name	Category	Description	Values	Default Value
relations_units_sensitive	Without category	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
remember_last_attach_type	Without category	Yes - Sets the default of the ATTACH TYPE menu of the last options picked.	yes, no	yes
remember_last_get_point_option	Drawing	Yes - Sets the default of the GET POINT menu as the last option picked among Pick Pnt, Vertex and On Entity.	yes, no	no
remember_replaced_components	Without category	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
rename_drawings_with_object	File Storage & Retrieval	Controls whether the system copies associated drawings automatically with parts and assemblies.	none, part, assem, both	none
replace_comp_name_method	Assembly	Sets the naming convention used when replacing components.	transfer, remove, preserve	transfer
replace_comp_notes_method	Without category	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
replace_history_rule_order	Assembly	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.	disable, 1, 2, 3, 4, 5	3
replace_interface_rule_order	Assembly	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.	disable, 1, 2, 3, 4, 5	1
replace_sameid_rule_order	Assembly	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.	disable, 1, 2, 3, 4, 5	5
replace_samename_rule_order	Assembly	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.	disable, 1, 2, 3, 4, 5	2

Name	Category	Description	Values	Default Value
replace_sameparam_rule_order	Assembly	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.	disable, 1, 2, 3, 4, 5	4
replace_unrelated_automatic	Assembly	Availability of automatic reference pairing in unrelated replace. No - Automatic reference pairing is not available. Yes - Automatic reference pairing is available.	yes, no	no
report_end_splice	Electromechanical	Setup whether the terminal splices of cables are reported.	yes, no	no
reserved_menu_space	User Interface	Reserves space for extra menus to the right of the Pro/ENGINEER window (0.0-4.0). This option overrides windows_scale.		1
restricted_gtol_dialog	Dimensions & Tolerances	Determines whether the Geometric Tolerance dialog restricts the user by graying out elements that are considered "illegal".	yes, no	yes
restricted_val_definition	Without category	Specify name and path of the file contained restricted values definition.		
retain_display_memory	File Storage & Retrieval	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
retrieve_data_sharing_ref_parts	File Storage & Retrieval	Automatic reference parts retrieval for dependent data sharing features.	yes, no, ignore_missing	no
retrieve_display	Without category	Controls whether or not graphics data is loaded in the background.	background, foreground	background
retrieve_instance_dependencies	Assembly	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
retrieve_merge_ref_parts	Without category	Automatic reference parts retrieval for dependent Merge.	yes, no, ignore_missing	no
reverse_spiral_finish_type	Without category	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.		

Name	Category	Description	Values	Default Value
ribbon_cable_flat_direction	Electromechanical	Flat ribbon cables in the y-axis of entry ports.	y_axis, x_axis	y_axis
right_mouse_button_popup	User Interface	Specify whether or not the right mouse button pop up menu is available or not.	yes, no	yes
room_rotation_lock	Photorender	Default room rotation setting at session start.studio - Lock room rotation to studio.model - Lock room rotation to model.	studio, model	studio
rotate_postscript_print	Printing & Plotting	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	
round_displayed_dim_values	Dimensions & Tolerances	No - newly created or first shown dimension is displayed with all its significant decimals."Rounded Dimension Value" Check Box will be unchecked while opening Dimension Properties Dialog. Yes - number of decimals is rounded according to default dec places option."Rounded Dimension Value" Check Box will be checked while openingDimension Properties Dialog.)	yes, no	yes
rv_current_object	Assembly	Will control default option when setting a model or component as the current object.	component, model, component + model	component
rv_search_scope	Assembly	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
save_bitmap	Data Management	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
save_bitmap_type	Data Management	When performing a save on a model, determines the type of image file that is created for that model. Used with "save_bitmap" option.	bmp, tiff, gif, cgm, jpeg	bmp
save_clipped_view	File Storage & Retrieval	Yes - clipping status is saved with model. No - clipping status is not saved.	yes, no	no
save_dialog_for_existing_models	File Storage & Retrieval	Controls whether the Save dialog is shown for models whose storage location is already known.	yes, no	yes

Name	Category	Description	Values	Default Value
save_display	Drawing	Yes - Stores view geometry and detail items such as solid dimensions. These items are displayed when retrieving the drawing in View-Only mode.	no, yes	no
save_display_status_in_submodel	Layers	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
save_drawing_picture_file	Drawing	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
save_hidden_items_w_status	Without category	The items on the "Hidden Items" temporary layer are stored permanently when layer display status is saved with Save Status command.	yes, no	yes
save_instance_accelerator	File Storage & Retrieval	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
save_model_display	File Storage & Retrieval	Sets amount of graphical data stored. Wireframe - Wireframe data. Shading_high - Most detail (shaded). Shading_low - Least detail (shaded). Shading_lod - Detail determined by ViewPerformance dialog (shaded).	wireframe, shading_low, shading_high, shading_lod	shading_lod
save_modified_draw_models_only	Drawing	Determines whether the system saves the model after you have changed it. No - Saves the model every time that you store the drawing.	no, yes	yes
save_object_in_current	File Storage & Retrieval	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
save_objects	File Storage & Retrieval	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

Name	Category	Description	Values	Default Value
save_scene_with_file	Photorender	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
save_texture_with_model	Photorender	Yes - Embed applied texture and tracesketch bitmap files into the model file.	yes, no	no
save_triangles_flag	Model Display	Spinning shaded images with software graphics only.Note: This option has no effect if your system uses hardware graphics.	no, yes	no
save_unchanged_pdm_object	Data Management	Determines how to save in the current working directory an object fetched from a Pro/PDM database into a Pro/ENGINEER session.	as_ref, as_copy	as_ref
saveunders_enabled	System	Yes - Enables Pro/ENGINEER 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
saving_model_thumbnails	File Storage & Retrieval	Enables saving of model thumbnails.	yes, no	yes
scope_invalid_refs	Without category	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
search_not_retrieved_models	Assembly	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
search_path	File Storage & Retrieval	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 Pro/ENGINEER's search path. Use full path name to avoid problems.		
search_path_file	File Storage & Retrieval	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 Pro/ENGINEER's search path. Use full path name to avoid problems.		

Name	Category	Description	Values	Default Value
section_color	Sketcher	Specifies the color of sketched sections.	default, sketch_color, drawing_color	
sel_insts_on_comp_retrieval	Assembly	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
select_hidden_edges_in_dwg	Drawing	Controls selectability of hidden edges in no-hidden drawing views.	yes, no	yes
select_on_dtm_edges	Environment	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
selection_of_removed_entities	Drawing	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
set_menu_width	User Interface	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
set_model_types_to_save	File Storage & Retrieval	Determines which model types are used in the Save Changed command. The format is a list of extensions separated by spaces.		asm
set_trail_single_step	System	Yes - Enables a trail file to be single-stepped by entering .	yes, no	no
shade_mfg_ref_models	Manufacturing	No - Does not process reference parts when generating the shaded image (to save time). Yes - Shades reference parts when generating shaded images.	no, yes	yes
shade_moving_comp	Model Display	Yes - Shades assembly components while they are being moved. No - Displays components in wireframe while they are being moved.	yes, no	yes

Name	Category	Description	Values	Default Value
shade_quality	Model Display	Model shade quality is used to decide how much to subdivide surfaces forshading purposes. Higher shade quality is slower but produces better pictures.	3, 1, 2, 4, 5, 6, 7, 8, 9, 10	3
shade_surface_feat	Model Display	Controls the shading of surface features.	yes, no	yes
shade_windows	Model Display	Enables shading of objects in all windows. This option does not also enable color shading.	all_windows, one_window	all_windows
shade_with	Model Display	Controls the display of datum curves on shaded objects.	curves, no	curves
shaded_postscript_background	Printing & Plotting	Specifies how background colors are handled in Postscript files of shaded objects.	yes, no	no
show_all_mold_layout_buttons	Casting & Mold Design	Sets all mold layout buttons visible.	yes, no	no
show_annotations_dialog_limit	Drawing	Default value for Annotations dialog box.		
show_axes_for_extr_arcs	Model Display	Determines if axes are created for newly extruded arcs.	yes, no	no
show_cav_fit_loc_sys	Miscellaneous	Allows the printing of the transformation matrix of a Pro/Verify Fit into a file.	yes, no	no
show_description_view_mgr	Without category	Yes - show a column for Description field for all tabs in view manager.	yes, no	no
show_dim_sign	Environment	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
show_geom_checks_on_creation	Features	Yes - The SHOW ERRORS menu appears at the end of feature creation when the feature being created has Geometry Checks.	yes, no	no
show_preview_default	Drawing	This option determines the default behavior for preview in Show/Erase.	keep, remove	remove
show_selected_item_id	Environment	Show id of Pro/E and Sketcher geometry in query selection bin, collector and tool tip.	no, yes	no
show_shaded_edges	Model Display	Controls the edge color when shading. Yes - Edges are displayed in darker color than the surfaces to which they belong. No - Edges are displayed in the same color as the surfaces to which they belong.	no, yes	no

Name	Category	Description	Values	Default Value
shrinkage_value_display	Casting & Mold Design	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
shrinkwrap_alert	Assembly	Displays the Shrinkwrap alert the first time the quality level is increased.	yes, no	yes
sim_additive_mass	Mechanica	Treats Mass Idealizations as additive if they are applied to the same entity and suppresses Mass precedence rules.	yes, no	no
sim_agem_edge_by_crv_defau lt	Mechanica	Sets the default value for the ratio of Edge Length / Radius of curvature.		2
sim_agem_edge_crv	Mechanica	Sets the value for the target ratio of Edge Length / Radius of curvaturefor automatic creation of a global AutoGEM Control.		0
sim_agem_model_thickness	Mechanica	Set the value to the part thickness. This causes AutoGEM to perform more boundary refinement, and can be very useful for thinner parts.		-1
sim_auto_map_max_angle	Mechanica	Sets the default maximum angle tolerance for auto-mapping of pressure load. The valid range is from 0 to 90 degrees.		10
sim_auto_map_max_dist	Mechanica	Sets the default maximum distance tolerance (relative to model size) for auto-mapping of pressure load. The default value is 0.0012.		0.0012
sim_auto_pairing_thickness	Mechanica	Set the value of characteristic thickness for auto-detection of shell pairs. Using this as a threshold value, auto-detection will find surface pairs with thickness smaller than this value.		0
sim_auto_regen_enabled	Mechanica	Yes - Regenerate model on entrance to Mechanica. No - Don't regenerate the model.	yes, no	yes
sim_beamsection_path	Mechanica	Specifies the path where the Beam Sections will be stored.		
sim_calculate_node_reaction	Mechanica	Enables node reaction calculations for point constraints within links. The default value is NO.	yes, no	no

Name	Category	Description	Values	Default Value
sim_check_jacobian_corners	Mechanica	No - Suppress Jacobian checks at corner nodes. ANSYS and Nastran checks parabolic elements Jacobian at the integration points only and FEM will fix too curved elements for the same criterion. Yes - Jacobian is checked both at integration points and corner nodes. FEM mode will fix too curved elements accordingly.	yes, no	no
sim_display_agem_controls	Mechanica	Toggles display of AutoGEM Controls.	yes, no	
sim_display_arrow_scale	Mechanica	Toggles automatic arrow scaling.	yes, no	
sim_display_arrow_tail_touching	Mechanica	Toggles display of Load arrows to have tails or heads touching.	yes, no	
sim_display_beam_releases	Mechanica	Toggles display of Beam Release icons.	yes, no	
sim_display_beam_sections	Mechanica	Toggles display of beam section icons.	yes, no	
sim_display_beams	Mechanica	Toggles display of beam icons.	no, yes	
sim_display_contacts	Mechanica	Toggles display of contact icons.	no, yes	
sim_display_current_csys_triad	Without category	Toggles display of the current coordinate system triad in the lower right corner of the window.	yes, no	
sim_display_fasteners	Mechanica	Toggles display of Fastener icons.	yes, no	
sim_display_highlight_refs	Mechanica	Yes -Geometrical references are highlighed when selecting a load/constraint.	yes, no	
sim_display_in_spin	Mechanica	Toggles display of simulation objects in Dynamic Spin.	yes, no	
sim_display_interfaces	Without category	Toggles display of structure connection icons.	no, yes	
sim_display_internal_mesh_edges	Mechanica	Specifies whether internal edges of mesh elements should be displayed in the wireframe mode. The default is "No".	yes, no	no
sim_display_load_colors	Mechanica	Toggles display of Load colors.	yes, no	
sim_display_load_distribution	Mechanica	Toggles display of distributed load vectors over entire entity.	yes, no	
sim_display_load_icons	Mechanica	Toggles display of load icons.	yes, no	
sim_display_load_name	Mechanica	Yes - The load/constraint name appears in the label.	yes, no	
sim_display_load_value	Mechanica	Toggles display of Load values.	yes, no	
sim_display_loads_from_comps	Mechanica	Toggles display of loads and constraints defined on lower-level components of assembly.	yes, no	
sim_display_masses	Mechanica	Toggles display of mass icons.	no, yes	

Name	Category	Description	Values	Default Value
sim_display_matl_assignments	Mechanica	Toggles display of Material Assignment icons.	yes, no	
sim_display_measures	Mechanica	Toggles display of Simulation Measure icons.	yes, no	
sim_display_mesh_and_model	Mechanica	Toggles display of Mesh together with Model.	yes, no	
sim_display_mesh_control_name	Mechanica	Yes - The Mesh/AutoGEM control name appears in the label.	yes, no	
sim_display_mesh_control_value	Mechanica	Yes - The Mesh/AutoGEM control value appears in the label.	yes, no	
sim_display_mesh_controls	Mechanica	Toggles display of Mesh Controls.	yes, no	
sim_display_mesh_entities	Mechanica	Toggles display of Mesh entities.	yes, no	
sim_display_mesh_mode	Mechanica	Specifies the mode of mesh display in FEM preprocessor. Values: NO_MESH, WIREFRAME, HIDDEN, NOHIDDEN, SHADING.	no_mesh, wireframe, hidden, nohidden, shading	
sim_display_mesh_quality	Mechanica	Specifies the quality of mesh display in Mechanica. Values: FINE, MEDIUM, COARSE.	fine, medium, coarse	
sim_display_mesh_shells_thick	Mechanica	Toggles the display of shell thickness in Mechanica mesh. YES - Display shells with real thickness. NO - Display shells with zero thickness.	no, yes	
sim_display_mesh_shrink_elems	Mechanica	Controls display of Mesh through a specified percentage of shrinking of its elements.		-1
sim_display_model_type	Mechanica	Toggles the display of the Simulation model setup in the lower left corner of Pro/E screen.	yes, no	yes
sim_display_names	Mechanica	Toggles display of Simulation Object names.	yes, no	
sim_display_rigid_dof	Mechanica	Toggles display of Rigid Link DOF icons.	yes, no	
sim_display_rigid_links	Mechanica	Toggles display of Rigid Link icons.	yes, no	
sim_display_shell_pairs	Without category	Toggles the display of shell pairs.	no, yes	
sim_display_shells	Mechanica	Toggles display of shell icons.	no, yes	
sim_display_springs	Mechanica	Toggles display of spring icons.	no, yes	
sim_display_struct_constraints	Mechanica	Toggles display of structure constraint icons.	yes, no	
sim_display_struct_loads	Mechanica	Toggles display of structure load icons.	yes, no	
sim_display_therm_bcs	Mechanica	Toggles display of Thermal Boundary Condition icons.	yes, no	
sim_display_therm_loads	Mechanica	Toggles display of thermal load icons.	yes, no	
sim_display_weighted_dof	Mechanica	Toggles display of Weighted Link DOF icons.	yes, no	
sim_display_weighted_links	Mechanica	Toggles display of Weighted Link icons.	yes, no	

Name	Category	Description	Values	Default Value
sim_display_welds	Mechanica	Toggles display of Weld icons.	yes, no	
sim_display_z_buffered	Without category	Toggles z-buffered display of Mechanica icons.	yes, no	
sim_expanded_diagnostics	Mechanica	Yes - Mechanica will perform an expanded set of diagnostic checks during meshing a model or running an analysis. These include checks for interferences in an assembly and for singular geometry. No - Mechanica will not perform expanded checks.	yes, no	no
sim_ext_distribution_dir	Without category	Specifies the location of external distribution files for loads.		
sim_extend_surf_max_angle	Mechanica	Controls the maximum angle between a surface to be extended during shell compression and its adjacent surfaces. If the angle is higher than specified, the adjacent surfaces is extended. The default is 30 degrees. The range is from 0 to 90 degrees.		30
sim_fatigue_biaxiality_correct	Mechanica	Controls the use of biaxiality correction. The default value is YES.	yes, no	yes
sim_fatigue_biaxiality_method	Mechanica	Controls which method is used to model biaxiality: Klann-Tipton-Cordes, Hoffman-Seeger, or the most conservative (worst) of the two methods. The default value is WORST.	worst, ktc, hs	worst
sim_fatigue_confidence_level	Mechanica	Specifies the percentage confidence in the predicted life result value. The default value is 90, and range is from 0.1 to 99.9		90
sim_fatigue_external_matdata	Mechanica	Controls the use of external material data. The default value is NO.	no, yes	no
sim_fatigue_hysteresis_gate	Mechanica	Specifies the gate applied to cycle counting as a percentage of peak load. The default value is 1 and range is from 0 to 50.		1
sim_fatigue_infinite_life_value	Mechanica	Specifies the value for the lives beyond cutoff. The default value is 1e20 and range is from 1e15 to 1e30.		1E+20
sim_fatigue_mean_stress	Mechanica	Controls the application of the mean stress correction. The default value is YES.	yes, no	yes
sim_fatigue_mean_stress_method	Mechanica	Controls which method is used to model mean stress: the Smith-Watson-Topper Approach, Morrow Correction, or the most conservative (worst) of the two methods. The default value is WORST.	worst, swt, morrow	worst

Name	Category	Description	Values	Default Value
sim_fatigue_safety_margin	Mechanica	Specifies a factor used to determine the Life Confidence quantity. The default value is 3 and range is from 1.1 to 100.		3
sim_fatigue_user_directory	Mechanica	Specifies the directory for user files (for example, external material files). The default value is the current directory.		
sim_fatigue_write_surf_stress	Mechanica	Controls writing of surface stresses to a neutral file. The default value is NO.	no, yes	no
sim_fem_nastran_use_pshell	Mechanica	Yes - use PSHELL cards, No - use PCOMP cards for laminate composite shells in NASTRAN output file.	yes, no	no
sim_html_report_prefs_file	Mechanica	Specifies the path to the Simulation post processor HTML report preferences file.		
sim_hyperelastic_material_fit	Mechanica	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
sim_icon_distribution_density	Mechanica	Determines how many icons are drawn for a distribution. 1 is fewer, 10 is the highest.		-1
sim_ignore_unpaired	Mechanica	Yes - Unpaired surfaces are ignored when meshing the model, placing you directly into the AFTER COMP menu (the CONT MESH menu is skipped).	yes, no	no
sim_invoke_diagnostics_on_run	Mechanica	Sets the default for the display of diagnostics during the run of an analysis in Mechanica Native mode. If set to prompt, the user is prompted to rundiagnostics every time he/she starts a run.	yes, no, prompt	prompt
sim_load_mech_mesh	Mechanica	No - The user is prompted to load the existing Mechanica mesh file from the current working directory. Yes - The mesh file, if exists, is loaded automatically and the mesh is displayed.	no, yes	no
sim_loadpreview	Mechanica	Sets the type of load preview: vectors, fringe or mesh.	vectors, fringe, mesh	fringe
sim_mat_poissons_notation	Mechanica	Specifies the convention used for defining Poisson's ratio for anisotropic materials as being either Tsai (Column-normalized) or Jones (Row-normalized). This option affects only the labels for Poisson's ratio on the Material data form.	tsai, jones	tsai
sim_max_contact_iterations	Mechanica	Specify the maximum number of iterations for contact analysis. The default is 200.		200

Name	Category	Description	Values	Default Value
sim_max_memory_usage	Mechanica	Specifies the maximum amount of memory in megabytes to be allocated during FEM meshing. If this option is not set, or set to zero, the mesher can use all available memory.		
sim_nastran_use_coupmass	Without category	Add PARAM,COUPMASS,1 in the Nastran deck file to enforce consistent mass matrix generation.	yes, no	no
sim_output_ids_for_layers	Mechanica	Outputs FEM mesh element IDs on each layer into XML file.	yes, no	no
sim_output_obj_names	Without category	Output Beam Sections names, Analyses names, Coordinate Systems names as comments.	yes, no	no
sim_pp_background_color	Mechanica	Specifies the background color of the work area for the results display. The default value is PROE.	proe, blue, black, white	proe
sim_pp_display_beam_thickness	Mechanica	Sets the thickness for beams and 2D-shells display in Post Processor. Numerical values specify the thickness in pixels.	actual, 1, 2, 3, 4, 5	5
sim_pp_display_spin_center	Mechanica	Sets the default for the behavior of mouse spinning the model in results.	yes, no	yes
sim_pp_legacy_selection	Mechanica	Enable legacy selection of geometrical entities from Mechanica postprocessor on Mechanica geometry.	no, yes	no
sim_pp_legend_continuous	Mechanica	Determines if the legend will be continuous tone (YES) or discrete colors (NO).	yes, no	no
sim_pp_legend_levels	Mechanica	The default number of legend levels (from 1 to 16; the default is 9).		9
sim_pp_legend_show	Mechanica	If set to NO, the legend is not displayed in newly created results windows by default.	yes, no	yes
sim_pp_legend_show_min_max	Mechanica	If set to NO, the gray minimum and maximum values are not displayed by default in the legend.	yes, no	yes
sim_pp_legend_spectrum	Mechanica	This string sets the default color legend used in the definition of a results window. This file will be looked up in the sim_pp_template_dir location, or, if this is not specified, in the current working directory.		
sim_pp_path_absolute	Mechanica	Choose Absolute (Yes) or Relative (No) path to "rwd" file.	yes, no	no
sim_pp_query_location	Mechanica	Sets the default value for the display of the WCS location in dynamic query in results.	yes, no	no
sim_pp_show_cap_cut_outline	Mechanica	Enables to hide the model outline in capping/cutting surface display.	yes, no	yes
sim_pp_template_dir	Mechanica	Sets the default directory for results template files.		

Name	Category	Description	Values	Default Value
sim_pp_vector_plot_arrow_2d	Mechanica	To draw 2D arrows instead of 3D arrows while post-processing vector results. It is useful when system memory is not sufficient to display3D vector arrows.	yes, no	no
sim_pp_vrml_export_format	Mechanica	Flags to denote VRML1.0 or VRML2.0 (default) output.	vrml2.0, vrml1.0	vrml2.0
sim_pp_vrml_feature_edges	Mechanica	Flag to enable the inclusion of feature edges in the VRML file. The default value is NO.	no, yes	no
sim_reentrant_geom_angle	Mechanica	Sets the default value for reentrant geometry detection in both Isolation for Exclusion AutoGEM Control and expanded diagnostics checks. The valid range is from 0 to 180 degrees. The default is 120 degrees.		120
sim_remove_unopposed	Mechanica	Yes - The system ignores unopposed surfaces when meshing the model; the system also deselects the UseUnopposed option. No - the system uses the unopposed surfaces in the shell model.	yes, no	no
sim_run_num_threads	Without category	Controls the number of CPUs that Mechanica Native mode will use for running the analysis. The possible values are: ALL (default), ALL_BUT_1, ALL_BUT_2, 2, 1, HALF.	all, all_but_1, all_but_2, 2, 1, half	all
sim_run_out_dir	Mechanica	Specifies the directory for Mechanica run output storage. By default, the current directory is used.		
sim_run_tmp_dir	Mechanica	Specifies the directory for Mechanica run temporary storage. By default, the current directory is used.		
sim_smooth_aspect_ratio	Mechanica	Sets the element aspect ratio criterion for the mesh smoothing process.		7
sim_smooth_edge_angle	Mechanica	Sets the element edge angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.		30
sim_smooth_skew	Mechanica	Sets the element skew angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.		45
sim_smooth_taper	Mechanica	Sets the element taper criterion (Quad elements only) for the mesh smoothing process.		0.5
sim_smooth_warp_angle	Mechanica	Sets the element warp angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.		10

Name	Category	Description	Values	Default Value
sim_solver_memory_allocation	Mechanica	Specifies the amount of memory in megabytes reserved for solving equations and for storing element data created by the iterative solver. The default value is 128.		128
sim_spotweld_split_mesh	Mechanica	If set to YES (default), the elements that touch the end of the spot weld are created in a circle the same diameter as the weld (AutoGEM only).	yes, no	yes
sim_struct_interface_default	Mechanica	Sets the default for the interfaces created between the geometry in a structural model for meshing and running analysis.	bonded, free, contact	
sim_therm_interface_default	Mechanica	Sets the default for the interfaces created between the geometry in a thermal model for meshing and running analysis.	bonded, adiabatic, resistance	
sim_use_layers	Mechanica	Allows layers to become groups for Mechanica Independent mode.	no, yes	no
simprep_default_model_status	Assembly	Determines the default model status while defining a new simplified rep.	exclude, master, graphics, geometry	exclude
simprep_ondemand_activation	Without category	Enables retrieving on-demand the Master rep ofan activated model. When an assembly is activatedthe Assembly only rep is retrieved.	yes, no	yes
simprep_ondemand_cleanup	Without category	Determines if a model retrieved on demandshould be removed or erased.Disabled - the model will not be removedand will remain in the assembly.Removed - the model will be removed. Remove_Erase - The model will be removedand erased from memory.	disable, remove, remove_and_erase	remove
simprep_ondemand_editing	Without category	Enables on demand retrieval of an edited item and its parent references.Configuration tools (Family Tables, Relations, etc) will retrieverequired models accordingly.Automatic - Retrieves the minimum rep required to perform the operation.	disable, master, automatic, master_with_ancestors	automatic
simprep_ondemand_regeneration	Without category	Enables on demand retrieval of requiredmodels during regeneration.Can affect regeneration performance.	automatic, master, disable	automatic
simprep_ondemand_selection	Assembly	Determines the rep to be retrieved when selecting a reference. AutomaticRetrieves the minimum rep required to perform the operation.	automatic, master, disable	automatic

Name	Category	Description	Values	Default Value
simprep_ondemand_settings	Assembly	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, never_prompt, disable	never_prompt
simulation_fem_mode	Mechanica	Yes - Activates FEM mode. No - Allows access to Mechanica only. Prompt (default) - Brings Mechanica Model Type Dialog. This option is used only if information about FEM mode is not yet stored with model.	yes, no, prompt	prompt
simulation_process_auto_advance	Mechanica	If set to yes the Process Guide will advance to the next set automatically upon completion of the current step.	yes, no	no
simulation_process_prompt	Mechanica	If set to yes, the Process Guide will be launched automatically upon entry into Mechanica, if template files are present.	yes, no	no
simulation_process_template_dir	Mechanica	Specifies the default location of the Process Guide template files.		
simulation_product	Mechanica	Set Mechanica product being selected (Structure/Thermal). Prompt (default) -Brings Mechanica Model Type Dialog. This option is used only if the information about Mechanica product is not yet stored with model.	structure, thermal, prompt	prompt
skeleton_model_default_color	Assembly	Specifies the color Pro/ENGINEER 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
sket_import_geometry_line_style	Sketcher	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_animated_modify	Sketcher	No - The section is not animated as modifications are regenerated.	yes, no	yes

Name	Category	Description	Values	Default Value
sketcher_auto_create_refs	Sketcher	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
sketcher_blended_background	Sketcher	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.	no, yes	no
sketcher_collinear_constr	Sketcher	Yes - collinear constraints will be used by Intent Manager; No - collinear constraints will not be used by Intent Manager.	no, yes	yes
sketcher_default_font_kerning	Sketcher	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_dim_of_revolve_axis	Sketcher	If this option is set all dimensions created by Intent Manager to Axis of Revolution will be diameter dimensions.	no, yes	yes
sketcher_dimension_autolock	Sketcher	Automatically lock strong sketcher dimensions.	no, yes	no
sketcher_disp_constraints	Sketcher	Shows the constraints when sketching in Sketcher Mode; for example, H for horizontal, V for vertical, and so on.	yes, no	yes
sketcher_disp_dimensions	Sketcher	No - Suppresses the display of all dimensions while in sketcher.	yes, no	yes
sketcher_disp_grid	Sketcher	Specify whether or not to display the sketcher grid.	yes, no	no
sketcher_disp_vertices	Sketcher	No - Suppresses the display of yellow points on all vertices while in sketcher.	yes, no	yes
sketcher_disp_weak_dimensions	Sketcher	Yes - weak dimensions will be displayed; No - weak dimensions will not be displayed.	yes, no	yes
sketcher_equal_length_constr	Sketcher	Yes - equal length constraints will be used by Intent Manager; No - equal length constraints will not be used by Intent Manager.	no, yes	yes
sketcher_equal_radii_constr	Sketcher	Yes - equal radii constraints will be used by Intent Manager; No - equal radii constraints will not be used by Intent Manager.	no, yes	yes
sketcher_grid_angle	Sketcher	Enter a grid angle value to override the default grid angle value.		0

Name	Category	Description	Values	Default Value
sketcher_import_exact_geom	Sketcher	During import in sketcher use exact geometry.	yes, no	yes
sketcher_lineup_hor_constr	Sketcher	Yes - line up horizontal constraints will be used by Intent Manager; No - line up horizontal constraints will not be used by Intent Manager.	no, yes	yes
sketcher_lineup_ver_constr	Sketcher	Yes - line up vertical constraints will be used by Intent Manager; No - line up vertical constraints will not be used by Intent Manager.	no, yes	yes
sketcher_lock_modified_dims	Sketcher	Yes - modified dimensions will be locked; No - modified dimensions will not be locked.	yes, no	no
sketcher_midpoint_constr	Sketcher	Yes - midpoint constraints will be used by Intent Manager; No - midpoint constraints will not be used by Intent Manager.	no, yes	yes
sketcher_palette_path	Sketcher	Sets the default directory for the user sketcher shape library. Use the full path name to avoid problems.		
sketcher_parallel_constr	Sketcher	Yes - parallel constraints will be used by Intent Manager; No - parallel constraints will not be used by Intent Manager.	no, yes	yes
sketcher_perpendicular_constr	Sketcher	Yes - perpendicular constraints will be used by Intent Manager; No - perpendicular constraints will not be used by Intent Manager.	no, yes	yes
sketcher_point_on_entity_constr	Sketcher	Yes - point on entity constraints will be used by Intent Manager; No - point on entity constraints will not be used by Intent Manager.	no, yes	yes
sketcher_refit_after_dim_modify	Sketcher	Refits section after dimension modification in 2D section or when creating the first feature.	yes, no	yes
sketcher_rel_accuracy	Sketcher	Enter a sketcher relative accuracy.		-1
sketcher_same_point_constr	Sketcher	Yes - same points constraints will be used by Intent Manager; No - same points constraints will not be used by Intent Manager.	no, yes	yes
sketcher_save_preview_image	Sketcher	This option controls whether section files should be saved with embedded image information which can be used to preview sections in File/Open dialog.	no, yes	no
sketcher_set_grid_method	Sketcher	"Automatic" - grid spacing will be automatically set, "Manual" - grid spacing will be manually set.	automatic, manual	automatic
sketcher_set_grid_x_spacing	Sketcher	Enter an x-grid spacing value to override the default x-grid spacing value.		1

Name	Category	Description	Values	Default Value
sketcher_set_grid_y_spacing	Sketcher	Enter a y-grid spacing value to override the default y-grid spacing value.		1
sketcher_starts_in_2d	Sketcher	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	yes
sketcher_strngthn_to_def_dec_pl	Without category	This option enable the user to keep a highly precise value during creation or convertion strong dimension.	yes, no	yes
sketcher_symmetric_constr	Sketcher	Yes - symmetric constraints will be used by Intent Manager; No - symmetric constraints will not be used by Intent Manager.	no, yes	yes
sketcher_tangent_constr	Sketcher	Yes - tangent constraints will be used by Intent Manager; No - tangent constraints will not be used by Intent Manager.	no, yes	yes
sketcher_undo_reorient_view	Sketcher	Controls the ability to Undo view reorientation while in Sketcher. Yes - it is possible to Undo view reorientation in Sketcher.	no, yes	no
sketcher_undo_stack_limit	Sketcher	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
skip_small_surfaces	Model Display	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.	no, yes	yes
smooth_lines	Model Display	Choose whether to smooth lines by default or not.	no, yes	no
smt_allow_flip_sketch	Sheetmetal	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
smt_bend_notes_dflt_display	Sheetmetal	Define the default state of Bend Note display (sheetmetal Mode)	yes, no	yes
smt_bend_notes_direction_down	Sheetmetal	Define the Down Direction symbol for Bend Notes (sheetmetal Mode)		default
smt_bend_notes_direction_up	Sheetmetal	Define the Up Direction symbol for Bend Notes (sheetmetal Mode)		default
smt_bend_notes_order	Sheetmetal	Define the Fields Order for Bend Notes (sheetmetal Mode)		&type&direction&angle

Name	Category	Description	Values	Default Value
smt_bend_notes_type_formed	Sheetmetal	Define the Formed symbol for Bend Notes (sheetmetal Mode)		default
smt_bend_notes_type_rolled	Sheetmetal	Define the Rolled symbol for Bend Notes (sheetmetal Mode)		default
smt_crn_rel_display	Sheetmetal	Yes - Corner relief notes will be displayed. No - Corner relief notes will not be displayed.	no, yes	yes
smt_form_abort_on_fail_plc	Sheetmetal	Yes - will always fail the Sheetmetal Form feature if there is a problem with its placement .	yes, no	no
smt_mp_method	Sheetmetal	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.	mass, cg, both	cg
smt_outside_mold_lines	Sheetmetal	Yes - Outside Mold Lines will be created during Flat Pattern creation.	yes, no	no
snap_single_constraints	Without category	When dynamically dragging components during component placement, allow to snap to references one constraint at a time.	yes, no	yes
spherical_map_size	Model Display	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
spin_center_display	Environment	Determines whether or not the spin center symbol displays.	yes, no	yes
spin_rate_zoom_sensitive	Model Display	Allows slower spinning when zoomed in on a model.	yes, no	no
spin_with_notes	Model Display	Yes - 3D feature and model notes are displayed during dynamic spinning of a model.	yes, no	yes
spin_with_orientation_center	Without category	Display orientation center while reorienting.	yes, no	yes
spin_with_part_entities	Model Display	Yes - Datum features will be displayed during dynamic spinning of a model.	yes, no	no
spin_with_silhouettes	Model Display	Yes - Silhouette lines will be displayed during dynamic spinning of a model.	no, yes	no
start_appmgr	Environment	Determines whether or not the application manager starts when Pro/ENGINEER is started.	yes, no	yes
start_model_dir	File Storage & Retrieval	Provides the complete path to the directory containing start parts and assemblies.		

Name	Category	Description	Values	Default Value
startup_check_for_visualization	Environment	Determines whether or not the visualization components are checked for installation upon startup.	yes, no	yes
std_nastran_template	Mechanica	Sets the file path of a NASTRAN deck template. Use full path to avoid problems.		
step_appearance_layers_groups	Data Exchange	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
step_export_ap214_asm_def_mode	Data Exchange	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
step_export_dwg_views	Data Exchange	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
step_export_format	Data Exchange	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, ap214_cd, ap202_is, 203_is_ext, ap214_dis, ap209_dis, ap214_is, ap203_e2	ap203_is
store_tool_path_file	Manufacturing	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
style_auto_regen_curves	Without category	Yes - Enable autoregeneration of curves.	yes, no	yes
style_auto_regen_shaded_surface	Without category	Yes - Enable autoregeneration of shaded surfaces.	yes, no	yes
style_auto_regen_surface	Without category	Yes - Enable autoregeneration of surfaces.	yes, no	yes

Name	Category	Description	Values	Default Value
style_connection_arrow_scale	Without category	Set scale factor of surface connection arrow. Number should be greater then 0.		1
style_default_connections	Without category	Yes - Add connections to a newly created Style surface.	yes, no	yes
style_display_grid	Without category	Yes - Display grid for active datum plane.	yes, no	yes
style_grid_spacing	Without category	Set the number of lines to display on the grid. Number should be equal or greater than 2.		5
style_state_transparency	Without category	Set the value of transparency for components set to "Transparent" in style state. Value must be between 0 and 100.		70
style_surface_mesh_display	Without category	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
style_surface_mesh_quality	Without category	Sets the quality of the surface mesh. Values range is between 0.1 to 10.		1
summary_help	User Interface	Refers to the one-line help Messages that are either displayed or not displayed.	yes, no	no
suppress_appearance_message	Colors	Suppress the lightworks appearance conversion notification dialog.	yes, no	no
suppress_dlg_license_expired	User Interface	Suppresses "License expired: List of options and expiration days" dialog		7
suppress_license_loss_dialog	User Interface	Yes - Suppresses the 'Regained Floating License' dialog box. No - The 'Regained Floating License' dialog box will display.	yes, no	no
switch_dims_for_notes	Drawing	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
sym_leader_orient_move_text	Drawing	With Pro/DETAIL, automatically regroups weld symbol instances after you move the text.	yes, no	no
symbol_instance_palette_file	Drawing	Specifies the location of the symbol instance palette.		
symbol_palette_input	Drawing	Controls the display of the special symbol palette during note creation.	yes, no	yes

Name	Category	Description	Values	Default Value
system_background_color	Colors	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
system_colors_file	Colors	Specifies the system color file. Use the full path name to avoid problems.		
system_curves_color	Colors	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
system_dimmed_menu_color	Colors	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
system_edge_high_color	Colors	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
system_geometry_color	Colors	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
system_hidden_color	Colors	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

Name	Category	Description	Values	Default Value
system_highlight_color	Colors	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
system_iges_header_file	Data Exchange	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.		
system_letter_color	Colors	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
system_section_color	Colors	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
system_sheetmetal_color	Colors	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
system_volume_color	Colors	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
tablet_device_name	System	Specifies the name of a digitizing tablet.		
tan_angle_for_disp	Model Display	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
tangent_edge_display	Model Display	Determines how edges between tangent surfaces are displayed.	no, solid, centerline, phantom, dimmed	solid

Name	Category	Description	Values	Default Value
tbl_driven_tol_val_edit	Dimensions & Tolerances	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
template_designasm	File Storage & Retrieval	Specifies the designated template assembly. Use full path to avoid problems.		inlbs_asm_design.asm
template_drawing	File Storage & Retrieval	Specify the drawing used as the default drawing template.		c_drawing.drw
template_ecadasm	File Storage & Retrieval	Specify the model used as the default ECAD assembly template.		
template_ecadpart	File Storage & Retrieval	Specify the model used as the default ECAD part template.		
template_esrasm	Without category	Specifies the designated template ESR assembly. Use full path to avoid problems.		inlbs_asm_esr.asm
template_flat_harness	Electromechanical	Specify the model used as the default flattened Harness assembly template.		
template_harnesspart	Electromechanical	Specify the model used as the default Harness part template.		
template_mfgcast	File Storage & Retrieval	Specify the model used as the default manufacturing cast template.		inlbs_mfg_cast.asm
template_mfgcmm	File Storage & Retrieval	Specify the model used as the default manufacturing cmm template.		inlbs_mfg_cmm.asm
template_mfgemo	File Storage & Retrieval	Specify the model used as the default manufacturing expert machinist template.		inlbs_mfg_emo.asm
template_mfgmold	File Storage & Retrieval	Specify the model used as the default manufacturing mold template.		inlbs_mfg_mold.asm
template_mfgnc	File Storage & Retrieval	Specify the model used as the default manufacturing assembly template.		inlbs_mfg_nc.asm
template_mold_layout	File Storage & Retrieval	Specify the mold layout assembly used as a default template.		inlbs_mold_lay.asm
template_sheetmetalpart	File Storage & Retrieval	Specify the model used as the default sheetmetal part template.		inlbs_part_sheetmetal. prt
template_solidpart	File Storage & Retrieval	Specify the model used as the default part template.		inlbs_part_solid.prt

Name	Category	Description	Values	Default Value
terminal_command	User Interface	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.		
terminator_table_out_unicode	Electromechanical	YES - Unicode strings are exported in terminator table files, NO - Unicode strings are converted to ASCII characters for terminator table export.	yes, no	yes
texture	Model Display	Yes - Applied textures will be displayed when the model is shaded.	no, yes	yes
texture_search_path	Model Display	Appends a directory to the search path for textures.		
thermo_position_hint	User Interface	Enables you to position the thermometer-type scales whenever they appear, so that they do not overlap the Pro/ENGINEER windows (if space permits, for example, if windows have been scaled).	no_window_overlap, window_overlap	window_overlap
tiff_compression	Data Exchange	TIFF exports are done without compression.	none, g4, packbits, deflate	none
tiff_type	Data Exchange	Determines the type of tiff items are exported to variables refer to color sets	rgb, palette, grayscale, mono	rgb
todays_date_note_format	Drawing	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
tol_display	Dimensions & Tolerances	Displays dimensions with or without tolerances.	yes, no	no
tol_mode	Dimensions & Tolerances	Sets the tolerance mode of newly created dimensions.	nominal, limits, plusminus, plusminussym, plusminussym_super	limits
tol_num_digits_default_driven	Drawing	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

Name	Category	Description	Values	Default Value
tolerance_class	Dimensions & Tolerances	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.	fine, medium, coarse, very_coarse	
tolerance_standard	Dimensions & Tolerances	Sets the tolerance standard used when creating the model.	ansi, iso	ansi
tolerance_table_dir	Dimensions & Tolerances	Sets the default directory for user-defined tolerance tables for ISO-standard models. All Holes and Shafts tables overwrite existing tables when loaded.		
toolkit_registry_file	Application Programming Interfaces	Tells Pro/E the full path to which toolkit registry file to be used. This option replaces the R17 option prodevdat.		
tracesketch	Without category	Display tracesketches outside of Style feature	no, yes	yes
trail_delay	System	Sets a delay in seconds between trail file steps.		
trail_dir	System	Instructs Pro/ENGINEER to create its trail file in the specified directory rather than in the startup directory.		
train_file_path	System	Specifies the default directory for training files. The path name must start from the root of the directory, for example, /home/users/library/training.		
transparency	Model Display	Yes - Transparent colors will appear as such when the model is shaded.	no, yes	yes
travel_bounds_display	Without category	Enable travel bounds display.	yes, no	no
travel_bounds_transparency	Without category	Transparency value for machine bound display.		0.8
triangulate_filled_areas	Drawing	Subdivide filled areas into triangles (may affect memory usage and plot files).	yes, no	no
troubleshooter_show_info_gcheck	Features	When set to Yes this option will add a "Show info items" filter to the View panel in the Troubleshooter.	no, yes	no
ttf_handling_for_plot_files	Printing & Plotting	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
udf_af_name_postfix	Features	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

Name	Category	Description	Values	Default Value
ui_theme	Without category	Setup Pro/E Theme.Standard will result in the Wildfire Theme on UNIX and "following the Windows theme on Windows XP & Win2K. Wildfire - Will always use the Wildfire Theme regardless of the OS settings.	wildfire_5, wildfire, standard	wildfire_5
units_system_dic_file	Piping	Specifies the file containing the mapping for MCAT filename and units.		<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_units_system_dic_file.ptd
update_copy_geom_location	Assembly	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
update_harness_mfg_line_color	Electromechanical	Update cable line color for harness-mfg part created in release 2000i or earlier.	yes, no	no
update_pre_130_harness	Electromechanical	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
update_pre_16_cable_layers	Electromechanical	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
update_rep_refs	Assembly	"Yes" updates top assembly simplified reps for replacements during regeneration and upon replacement.	yes, no	yes
use_8_plotter_pens	Printing & Plotting	Specifies whether to support up to 8 plotter pens. The initial default is 4 pens.	no, yes	no
use_export_2d_dialog	Data Exchange	Yes - Opens export options dialog when exporting Pro/E drawing. No - exports file without the options dialog.	yes, no	yes
use_iges_font_1003	Data Exchange	Used to disable the use of IGES font 1003.	yes, no	yes
use_iges_kanji_font_2001	Data Exchange	Specifies whether to convert Kanji notes in Pro/ENGINEER to IGES Kanji notes (font code) when exporting. Yes - Converts Kanji notes to IGES Kanji notes. No - Converts using Font 1.	yes, no	no

Name	Category	Description	Values	Default Value
use_major_units	Dimensions & Tolerances	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
use_master_for_bad_disp_reps	Without category	Yes to retrieve master rep when graphics rep has no display data.	yes, no	no
use_nom_dim_val_in_expr	Dimensions & Tolerances	Yes - use dimensions' nominal values in expressions.No - use current values.	yes, no	no
use_part_color_for_hidden_lines	Colors	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.	no, yes	no
use_pre_wildfire_text_font	Model Display	Yes - Use the old stroke-based PTC font as the default font. No - Use a True Type font as the default font.	no, yes	no
use_software_linefonts	Printing & Plotting	Yes - The exact line style used in Pro/ENGINEER 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 Pro/ENGINEER.	yes, no	no
use_strict_ui_trail	Without category	Perform extra validation of user interface actions when playing back trail files.	yes, no	yes
use_temp_dir_for_inst	File Storage & Retrieval	Explicitly makes Pro/ENGINEER use the Temp directory for regenerating instances of models.	no, yes	no
use_web_page_font_in_reports	Without category	Controls whether HTML reports use the browser's standard fonts.	yes, no	no
user_defined_grid_interval	Features	Modify grid space in user defined length units for handle movement.		0.5
user_iges_header_file	Data Exchange	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.		
variable_plots_in_inches	Printing & Plotting	No - Variable plot sizes may be entered in millimeters.	yes, no	yes

Name	Category	Description	Values	Default Value
variant_drawing_item_sizes	Drawing	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
vda_header	Data Exchange	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.		
verify_on_save_by_default	File Storage & Retrieval	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
versatec_cutter_installed	Printing & Plotting	Yes - Indicates that a cutter is installed on a Versatec plotter.	no, yes	
visible_annotations_scope	Assembly	Specify whether sub-model annotations will be visible in assemblies.If set to "Active model only", only annotations of the active model will be visible.If set to "All", all annotations in sub-models will also be displayed. Annotation display in part mode is not affected.	active model only, all	active
visible_mapkeys	User Interface	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.	no, yes	no
visible_message_lines	User Interface	Sets the default number of visible message lines in the Pro/ENGINEER message area.		2
vrml_anchor_url	Data Exchange	Enables you to place an anchor on a specified VRML component during the export to VRML. The keywords are optional.		
vrml_background_color	Data Exchange	Yes - Exports a model to VRML with the Pro/ENGINEER background color.	yes, no	no
vrml_explode_lines	Data Exchange	Yes - Exports a model to VRML with explode lines for assembly or assembly process data.	yes, no	yes
vrml_export_resolution	Data Exchange	Specifies the number of levels of detail (LODs) in models exported in VRML format.	high, medium, low	medium

Name	Category	Description	Values	Default Value
vrml_export_version	Data Exchange	Allows the user to choose between exporting to the VRML 2.0 or 1.0 format.	2.0, 1.0	2
vrml_file_duplicate_material	Data Exchange	Yes - Ensures that the model components retain their true color. No - In some viewers, component colors might not be consistent.	yes, no	no
vrml_multiple_views	Data Exchange	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.	none, all, top	all
vrml_parameters	Data Exchange	Controls the export of user parameters. Designated - Export only designated parameters. All - Export all parameters. None - Do not export parameters.	designated, all, none	designated
vrml_simprep_export	Data Exchange	Yes - Specifies direct and in memory export of top-level assembly simplified representation to Pro/FLY-THROUGH package files.	yes, no	no
warn_if_iso_tol_missing	Without category	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
wcell_fixt_info_setup_file	Without category	Specify name and path of the file containing workcell and fixture name info.		
web_browser_history_days	System	Enter number of days to store history records.		20
web_browser_homepage	System	Enter location of Pro/E browser home page.		<proe_WF_5.0_loadpoint>/html/chinese_tw/proe/helpsystop/resource_center_index.htm
web_disable_js_command	Application Programming Interfaces	Enter a specific javascript command to be disabled		
web_enable_javascript	Without category	Enables or disables Pro/Web.Link in the embedded browser.	on, off	off
web_enable_subscription	Application Programming Interfaces	Enable the subscription mechanism for Connections	no, yes	yes
web_link_file_read	Application Programming Interfaces	Sets Pro/Web.Link permission to read data from the local file system.	yes, no	no

Name	Category	Description	Values	Default Value
web_link_file_write	Application Programming Interfaces	Sets Pro/Web.Link permission to write data to the local file system.	yes, no	no
web_link_proe_read	Application Programming Interfaces	Sets Pro/Web.Link permission to read data from a Pro/ENGINEER session.	yes, no	no
web_link_proe_write	Application Programming Interfaces	Sets Pro/Web.Link permission to modify data in a Pro/Engineer session.	yes, no	no
web_link_security_autoprompt	Application Programming Interfaces	Yes - The Security dialog box is displayed when a Pro/Web.Link function requires security access beyond the current settings.	yes, no	yes
web_max_mail_size	Without category	Set the maximum email attachment size in kilobytes.		5000
weld_ask_xsec_refs	Weld	No - Do not ask for X-section references when creating weld features. Yes - Ask for X-section references when creating weld features.	no, yes	no
weld_color	Weld	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
weld_dec_places	Weld	Sets the default number of decimal places (0-10) to display in weld parameters.		3
weld_edge_prep_driven_by	Weld	Determines if the edge preparation feature is created in the part or theassembly level.	part, assembly	part
weld_edge_prep_groove_angle	Weld	Specifies an initial default value for the angle cut edge preparation.		45
weld_edge_prep_groove_depth	Weld	Specifies an initial default value for the edge preparation depth.		0.25
weld_edge_prep_instance	Weld	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_edge_prep_name_suffix	Weld	Specifies the suffix name for the instance that will be created during edge preparation. Part name plus extension makes the instance name.		_noep
weld_edge_prep_root_open	Weld	Specifies an initial default value for the root opening edge preparation.		0.25

Name	Category	Description	Values	Default Value
weld_edge_prep_visibility	Weld	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_fillet_preferences_file	Without category	Sets the path and name of the fillet weld preferences file.		
weld_general_preferences_file	Without category	Sets the path and name of the general weld preferences file.		
weld_geom_type_default	Weld	Sets the default weld geometry type.	surface, light	surface
weld_groove_preferences_file	Without category	Sets the path and name of the groove weld preferences file.		
weld_notch_corner_radius	Weld	Specifies an initial default value for the radius at weld notch corner.		0.1
weld_notch_height	Weld	Specifies an initial default value for the weld notch height.		0.4
weld_notch_radius	Weld	Specifies an initial default value for the weld notch radius.		0.5
weld_notch_width	Weld	Specifies an initial default value for the weld notch width.		0.5
weld_plug_slot_preferences_file	Without category	Sets the path and name of the plug/slot weld preferences file.		
weld_spot_preferences_file	Without category	Sets the path and name of the spot weld preferences file.		
weld_ui_standard	Weld	Specifies standard for welding user interface. ANSI - Use ANSI standards. ISO - Use ISO standards.	iso, ansi	ansi
window_gamma	Photorender	Gamma correction to apply to images created by renderer. Default value is 1.0.		1
windows_browser_type	System	Select the type of embedded browser to use.	ie_in_process, ie_in_sep_process, mozilla_based_browser	
windows_scale	User Interface	Scales Pro/ENGINEER windows with a given coefficient. A value of 0.85 is usually adequate to allow dynamic menus to display to the right of the Pro/ENGINEER window.		1

Name	Category	Description	Values	Default Value
workpiece_instances	Without category	Instances are created in Workpiece for each steps.	steps, operations_and_steps, none, operations	none
www_add_aux_frame	Data Exchange	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.	no, yes	no
www_export_geometry_as	Data Exchange	Specifies the output format by to one of the values	vrml, cgm, jpg, cgm_vrml, jpg_vrml, cgm_jpg, all	jpg_vrml
www_multiple_views	Data Exchange	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.	none, all, top	top
www_tree_location	Data Exchange	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
x_angle	Model Display	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
y_angle	Model Display	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

Pro/ENGINEER Wildfire 4.0 to 5.0 Configuration Options Changes

Name	Status	Category	Description	Values	Default Value
acad_brk_tbls_w_blank_lines	DELETED	Data Exchange	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
accessory_window_display	NEW	Without category	Docked - Displays the accessory window embedded in the active window. Undocked - Displays the accessory window as a standalone window.	undocked, docked	docked
advanced_style_surface_edit	CHANGED	Features Without category	Use advanced options by default for Surface Edit in Style.	yes, no	no
allow_save_failed_model	NEW	Assembly	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
allow_update_part_simp_rep_data	CHANGED	Assembly	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
analysis_force_wireframe_mode	DELETED	Model Display	Force part to be displayed in wireframe mode if it has visible saved shaded analyses.	yes, no	no
animate_explode_states	NEW	Model Display	Animate the transition between explode states and mold opening steps.	yes, no	yes
auto_backup_new_placemnt_refs	NEW	Assembly	Yes - Create Component Reference Backup, No - Do not Create Component Reference Backup.	yes, no	no
auto_evaluate_simprep_rules	NEW	Assembly	If the option is "yes" newly created rep would be updated each time on retrieve/regeneration.	no, yes	no
auto_place_max_number	NEW	Assembly	Set the default value for the maximum number of matches allowed.		5
autohide_external_sketch	NEW	Features	If set to yes, a Sketch feature is automatically hidden after being used as external sketch for a sketch based feature.	yes, no	yes
boeing_extract_bundle	CHANGED	Without category	Setup whether the new extract bundle algorithm should be used.Yes - Uses the extract bundle algorithm. No - Does not use the extract bundle algorithm.	yes, no	no

Name	Status	Category	Description	Values	Default Value
cadam_line_weights	DELETED	Data Exchange	Define the line width of entities within Pro/ENGINEER to plot drawings with correct line weights that are consistent with a standard. The default Pro/ENGINEER values for these weights are: 2 (light), 3 (medium), 5 (heavy).		0
cdt_transfer_details	DELETED	Data Exchange	No - Details (dittos) associated with the imported CADAM drawing are placed on the current Pro/ENGINEER drawing sheet. Yes - Each detail (ditto) associated with the imported CADAM drawing converts to a separate additional sheet.	yes, no	yes
check_interface_criteria	NEW	Assembly	Check for interfaces when placing a component.	yes, no	no
cmm_custom_tool_param_file	NEW	Manufacturing	Defines the file that contains the user-defined tool parameters' list.		
combined_views_customization	NEW	Without category	Enable / Disable combined views customization.	yes, no	yes
comp_placement_assumptions	NEW	Assembly	Use placement assumptions when placing components.	yes, no	yes
compress_output_files	CHANGED	File Storage & Retrieval	You can compress object files to store them. Compressed files are generally somewhat smaller than non-compressed files (but may be larger in some cases). Compressed files are slower to read and write. Yes - Stores object files in compressed format.Yes - Stores object files in a compressed format.Note: Compressed files are usually smaller, buy may be larger,than uncompressed files. Compressed files are slower to read and to write to.	yes, no	no
contact_surfaces_gap	NEW	Without category	Sets the maximum allowable gap between weld contact surfaces.		0
datum_point_symbol	CHANGED	Model Display	Specifies the shape of datum point symbols displayed in Part or Assembly mode.	dot, cross, circle, triangle, square	cross
default_ang_dec_places	CHANGED	Drawing	Specifies the number of decimal places shown in angular dimensions in a drawing.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

Name	Status	Category	Description	Values	Default Value
default_ang_units	NEW	Dimensions & Tolerances	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
default_dec_places	CHANGED	Dimensions & Tolerances	Sets the default number of decimal places (0-14) to display in all model modes for non-angular dimensions. It does not affect the display of dimensions modified using Num Digits. In Sketcher, sketcher_dec_places controls the number of decimal 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
default_scene_filename	CHANGED	Photorender	Complete path to the scene file to be used as default scene file.		<proe_WF_5.0_loadpoint>/graphic-library/scenes//default-enhanced-realism.scn
display_mfg_icon_for_mfg_assy	NEW	Manufacturing	Yes- Display manufacturing icon for manufacturing assembly files in Open/Save dialogs.No- the manufacturing assembly files have assembly icons.	yes, no	no
display_pdm_server_quick_view	NEW	Without category	Yes - display summary view of PDM information for items in session. It applies to ProductPoint only.	yes, no	yes
display_popup_viewer	NEW	Without category	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
display_route_cable_preview	NEW	Electromechanical	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
display_z_extension_lines	CHANGED	Model Display	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
dm_nc_references	NEW	Without category	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

Name	Status	Category	Description	Values	Default Value
dm_user_presence	NEW	Data Management	Enable user presence indicator and actions.This capability requires a connection with a PDM server and requires an installed and running Microsoft communicator client.	yes, no	yes
drawing_setup_file	CHANGED	Drawing	Sets the default drawing setup file option values for your Pro/ENGINEER 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.		\<proe_WF_5.0_lo adpoint\>/text/prod etail.dtl
dwg_unicode_conversion_la nguage	CHANGED	Drawing	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 Pro/E session.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 Pro/ENGINEER session.	file, session, usascii, english, japanese, french, german, russian, italian, spanish, korean, chinese_tw, chinese_cn, hebrew, greek, turkish, czech, polish, hungarian,	file
dxf_export_mapping_file	DELETED	Data Exchange	Specifies the DXF export mapping file. Absolute or relative path to the file can be used.		
dxf_out_stroke_text	DELETED	Data Exchange	Strokes == text is converted to lines and dots. "NONE"- Does not stroke out the text for export to DXF or DWG. "ALL" - Strokes out all the text. "SPECIAL" - Strokes out only special characters.	none, special, all	none
dxfio_in_sjis	CHANGED	Data Exchange	Determines if the SJIS is supported when importing/exporting DXF files in Japanese	yes, no	yes
dynamic_edit_show_dims	NEW	Without category	Controls the display of dimensions during dynamic edit.	yes, no	yes
dynamic_edit_show_sketch_ dims	NEW	Without category	Controls the display of sketch dimensions during dynamic edit.	yes, no	yes
ecad_comp_xsec_def_name	CHANGED	Without category Data Exchange	Will allow pre-setting default X-section to be used for component outline output to *.eda format.		

Name	Status	Category	Description	Values	Default Value
ecad_mtg_hole_import	NEW	Data Exchange	Setup the default for the import of ECAD Holes type MTG (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
ecad_pin_hole_import	CHANGED	Data Exchange	Setup the default for the import of ECAD Holes from type PIN (IDF 3.0 ONLY)Set up the default for the import of ECAD Holes from type PIN (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	no
ecad_tool_hole_import	NEW	Data Exchange	Setup the default for the import of ECAD Holes type TOOL (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
ecad_via_hole_import	CHANGED	Data Exchange	Setup the default for the import of ECAD Holes from type VIA (IDF 3.0 ONLY)Set up the default for the import of ECAD Holes from type VIA (IDF 3.0 or IDX ONLY)	yes, no, as lightweight	yes
efx_enabled	NEW	Data Exchange	Loads EFX application.	yes, no	no
enable_3dmodelspace_browser_tab	NEW	Without category	Yes - Display 3D ModelSpace tab in the browser when Pro/ENGINEER starts.No - Display 3D ModelSpace tab only when you click a 3D ModelSpace link in the navigator.	yes, no	yes
explode_animation_max_time	NEW	Model Display	Sets the maximum amount of time for an animation between explode states.		1
extended_esr_capabilities	NEW	Assembly	Extends external simplified representation capabilities.	yes, no	yes
fem_shell_elem_type	NEW	Mechanica	Sets the default shell element type in FEM mode.	tri, quad	quad
file_open_default_view	CHANGED	Without category File Storage & Retrieval	Specify the initial Views setting as List or Details in the File dialog boxes.	list, details	list
flange_shape_sketches_directory	NEW	Sheetmetal	Sets path to directory that contains the Flange shapes.		
flat_shape_sketches_directory	NEW	Sheetmetal	Sets path to directory that contains the Flat shapes.		

Name	Status	Category	Description	Values	Default Value
frt_enabled	NEW	Data Exchange	Loads feature recognition application.	yes, no	no
geometry_analysis_color_scheme	NEW	File Storage & Retrieval	Specifies the location of the geometry analysis color scheme file.		
graphics	CHANGED	System	Sets the optional graphics environment used by certain platforms for running Pro/ENGINEER. Refer to the Installation Guide for a list of defaults and optional graphics.	opengl, x_windows, win32_gdi, d3d	opengl
help_center_path	NEW	User Interface	URL of an active Pro/ENGINEER Help Center, if the Help Center is not installed at the default path (along with Pro/E product), of nature http://localhost:/index.jspx. Or, URL of the Pro/ENGINEER Help Center, that is deployed on a web server.		
hlr_for_xhatches	CHANGED	Electromechanical	Controls whether HLR to be enabled for crosshatches in drawing mode.	no, yes	yes
idd_repair_tangency	NEW	Without category	Sets the default state of the "Repair Tangency" checkbox in the Repair and Match tools	yes, no	no
interface_criteria_mismatch	CHANGED	Without category	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.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.	warning_only, fail_placement, ignore	warning_only
intf2d_in_dxf_mapping_file	NEW	Data Exchange	Specifies the mapping file for DXF and DWG import. Absolute or relative path to the file can be used.		
intf2d_in_iges_symbols	NEW	Data Exchange	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

Name	Status	Category	Description	Values	Default Value
intf2d_in_open_log_window	NEW	Data Exchange	Yes - Opens Import Log File in a separate window. No - The Log File does not appear in the window.	yes, no	no
intf2d_out_acad_brk_tbl_w_bl_ln	DELETED	Data Exchange	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
intf2d_out_acad_unicode	NEW	Data Exchange	Yes - Non-ASCII characters are written in UNICODE encoding.	yes, no	no
intf2d_out_dxf_mapping_file	NEW	Data Exchange	Specifies the mapping file for DXF and DWG export. Absolute or relative path to the file can be used.		
intf2d_out_stroke_text	CHANGED	Without category Data Exchange	make stroke text as default for PDF exportStrokes == 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.	none, special, all	none
intf3d_out_annotations	NEW	Data Exchange	Export annotations with models.	yes, no, default	yes
intf3d_out_export_as_facets	DELETED	Data Exchange	Export as faceted data for formats that support both facet and Brep representations.	facets_only, brep_and_facets, default	default
intf3d_out_jt_auto_lods	NEW	Data Exchange	Generate three default ProE LODs.	no, yes	no
intf3d_out_jt_brep	NEW	Data Exchange	Export as faceted data for formats that support both facet and Brep representations.	no, jt_brep, xt_brep	no
intf3d_out_jt_config_name	NEW	Without category	Name of JT configuration file.		
intf3d_out_jt_config_path	NEW	Without category	Path to JT configuration file.		
intf3d_out_parameters	CHANGED	Without category Data Exchange	Exports parameters with models.	all, none, designated	all

Name	Status	Category	Description	Values	Default Value
intf3d_out_prop_chord_heights	CHANGED	Data Exchange Without category	Use maximum chord heights proportional to component sizes.	yes, no	no
intf3d_out_prop_step_sizes	CHANGED	Data Exchange Without category	Use maximum step sizes proportional to component sizes.	yes, no	no
intf3d_out_use_step_size	CHANGED	Data Exchange Without category	Use maximum step size(s).	yes, no	no
intf_generate_log_file	NEW	Data Exchange	Specifies not to generate or generate short or long log files for data exchange import operations.	no, short, long	short
intf_in_profile_default	NEW	Data Exchange	Specifies the name of the existing import profile to use by default.		
intf_profile_dir	NEW	Data Exchange	Specifies the directory where import and export profiles are stored. Use the full path to avoid problems.		
intf_ug_version	CHANGED	Data Exchange	Sets the version of Unigraphics interface.	nx3, nx4, nx5	nx4
isogen_attribute_map_file	CHANGED	Piping	Specifies the mapping between isogen parameter and attribute name.		<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_attribute_map.ptd
isogen_endtype_map_file	CHANGED	Piping	Specify name of file mapping Pro/Piping endtypes with ISOGEN endtypes		<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_endtype_map.ptd
isogen_nominal_size_map_file	CHANGED	Piping	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches		<proe_WF_5.0_loadpoint>/text/piping_data/isodata/isogen_nominal_size_map.ptd

Name	Status	Category	Description	Values	Default Value
isogen_symbol_map_file	CHANGED	Piping	Specify name of file for mapping Pro/Piping selection names with ISOGEN symbol keys (SKEY)s		<proe_WF_5.0_lo adpoint>/text/pipin g_data/isodata/iso gen_symbol_map .ptd
keep_model_in_open_windo w	NEW	Without category	Controls whether a model is displayed in its window whenthe accessory window is open.	no, yes	no
keep_style_items_hidden_o n_exit	CHANGED	Features Without category	Keep style entities hidden after exiting of the Style feature definition.	yes, no	no
last_session_directory_path	NEW	Without category	Sets the directory that holds the temp files for the session recording mechanism.		/work_dir/.wf/.setti ngs
last_session_retrieval	NEW	Without category	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
make_aux_view_notes	CHANGED	Drawing	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
manikin_flexibility	CHANGED	Without category	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.	full, none, default	default
mdo_density	DELETED	Mechanism	Mass density for dynamics.		0
mdx_belt_icon_width	NEW	Mechanism	Width of belt transmission icons.		5
menu_manager_position	NEW	User Interface	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.	outside, inside, adaptive	adaptive
merge_type	NEW	Features	Sets the default merge type surface merge operations.	join, intersect	intersect
modeltree_show_gen_of_ins tance	NEW	User Interface	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

Name	Status	Category	Description	Values	Default Value
move_mechanism_in_nongeom_reps	NEW	Mechanism	Allow mechanism operations in non geometrical representations.	no, yes	yes
multipoint_location_count	DELETED	Electromechanical	Specifies the maximum number of cable locations to allow in one feature. To create locations as features, set to 1.To limit the number of locations in one feature, set to a number between 2 and 100.		1
nc_autodoc_param_dir	NEW	Manufacturing	Location of the setup file that describes which process parameters are included in the parameter table.		
nc_autodoc_report_dir	NEW	Manufacturing	Location for storing the report and images generated by the shop floor documentation tool.		
nc_autodoc_setup_dir	NEW	Manufacturing	Location for documentation setup files that describe which process parameters are included in the parameter table.		
nc_autodoc_template_dir	NEW	Manufacturing	Location of the style sheet and default images used for shop floor documentation.		
ncmdl_bar_stock_part_path	CHANGED	Manufacturing	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<proe_WF_5.0_loadpoint>/text/ncmdl_data/ncmdl_bar.prt
ncmdl_billet_stock_part_path	CHANGED	Manufacturing	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.		<proe_WF_5.0_loadpoint>/text/ncmdl_data/ncmdl_billet.prt
new_wf5_simp_rep_ui	NEW	Without category	Yes - Use Component Chooser base Simp Rep UI, No - Use old Simp Rep definition UI.	yes, no	yes
pdm_server_quickview_rows_num	NEW	Without category	Sets the default number of rows for the pdm quick view list.		7
pipeline_assembly_library_dir	CHANGED	Piping	Sets the default pipeline assembly library directory.		<proe_WF_5.0_loadpoint>/text/piping_data/pipelinelib/
piping_appearance_map_file	CHANGED	Piping	Sets the default piping appearance map filename.		<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_appearance.map

Name	Status	Category	Description	Values	Default Value
piping_bolt_nut_select_file	CHANGED	Piping	Piping bolts and nuts selection file.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/master_ca talog/piping_bolt_ nut_select.ptd
piping_end_compatibility_file	CHANGED	Piping	Sets the default piping end compatibility filename.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/master_ca talog/piping_end_ compatibility.ptd
piping_extend_dim_scheme	CHANGED	Piping	Specify a dimension scheme for piping extend segment.	cartesian, cylindrical,	cartesian
piping_extend_show_csys	NEW	Piping	Control whether small preview CSYS is shown at end of pipe when defining extend features in cartesian / spherical / cylindrical schemes.	yes, no	no
piping_fitt_category_map_fil e	CHANGED	Piping	Sets the default piping fitting category map filename.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/sample_pr oject/piping_fitt_c ategory_map.ptd
piping_fitt_library_dir	CHANGED	Piping	Sets the default piping fitting library directory.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/fittinglib/
piping_insulation_dir_file	CHANGED	Piping	Sets the default piping insulation directory filename.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/sample_pr oject/piping_insul ation_dir.ptd
piping_manufacture_dir_file	CHANGED	Piping	Sets the default piping manufacture directory filename.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/master_ca talog/piping_man ufacture_dir.ptd
piping_material_file	CHANGED	Piping	Sets the default piping material filename.		\<proe_WF_5.0_lo adpoint\>/text/pipin g_data/master_ca talog/piping_mate rial.ptd

Name	Status	Category	Description	Values	Default Value
piping_mcat_dir	CHANGED	Piping	Sets the default piping master catalog directory.		\<proe_WF_4.0_loadpoint>/text/piping_data/master_catalog/\<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/
piping_mcat_dir_file	CHANGED	Piping	Sets the default piping master catalog directory filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_mcat_dir.ptd
piping_project_data_dir	CHANGED	Piping	Sets the default Project data directory.		/\<proe_WF_5.0_loadpoint>/text/piping_data/sample_project/
piping_spec_dir_file	CHANGED	Piping	Sets the default piping specification directory filename.		\<proe_WF_5.0_loadpoint>/text/piping_data/sample_project/piping_spec_dir.ptd
preferred_publish_deliverable	NEW	Drawing	Set the default preferred deliverable for the Publish Context in Drawings.	print, iges, set, dxf, step, cgm, pdf, dwg, tiff, medusa,	print
pro_font_dir	CHANGED	File Storage & Retrieval	Sets the default font directory (Pro/ENGINEER will load all fonts in this directory into the drawing regardless of the contents of the system and working directories).		\<proe_WF_5.0_loadpoint>/text/fonts/
pro_gplug_dir	CHANGED	Data Exchange	Specifies the directory in which Granite application plug-ins are located. Use the full pathname to avoid problems.		\<proe_WF_5.0_loadpoint>/sun_solaris_x64/gplugs/
pro_lwa_library_path	DELETED	Without category	Allows the user to specify the path to LWA library.		\<proe_WF_4.0_loadpoint>/graphic-library/smdata/archives/
pro_lwa_license_path	DELETED	Photorender	Allows the user to specify the path to LWA license file.		
pro_mf_tape_dir	NEW	Manufacturing	Sets the default directory for the manufacturing TAPE data files.Use the full pathname to avoid problems.		

Name	Status	Category	Description	Values	Default Value
pro_palette_dir	CHANGED	Drawing	Sets the default directory for drawing symbol palette files.		\<proe_WF_5.0_lo adpoint\>/symbols/ palette/
pro_surface_finish_dir	CHANGED	File Storage & Retrieval	Sets the default directory for your user-defined surface finish symbols. Use the full pathname to avoid problems.		\<proe_WF_5.0_lo adpoint\>/symbols/ surffins/
ptc_manikin_library_path	CHANGED	Without category	Sets the default directory for the Pro/ENGINEER manikin library (or custom manikin libraries you built containing the proper index menu files). Use the full path name to avoid problems.		\<proe_WF_5.0_lo adpoint\>/manikin _lite/
punch_models_directory	NEW	Sheetmetal	Sets path to directory that contains the punch models.		
read_parameters_of_exclud ed	CHANGED	Data Management	Allow Pro/ENGINEER designated parameters on excluded objects in Simplified Representation to be visible in Model Tree columns.	yes, no	no
real_time_rendering_display	NEW	Model Display	Disable realtime rendering.	disable, transparent_floor, room_display	disable
regen_failure_handling	NEW	Without category	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
regen_simp_retrieve	CHANGED	File Storage & Retrieval	Regenerate the assembly when retrieving an assembly simplified representation. Yes - Regenerates the assembly on retreival.No - The assembly is not regenerated on retrieval.	yes, no	yes no
regen_solid_before_save	DELETED	Without category	Yes - means to regenerate top Window object before Save.	yes, no	no
replace_history_rule_order	CHANGED	Assembly	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.	disable, 1, 2, 3, 4, 5	3
replace_interface_rule_order	CHANGED	Assembly	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.	disable, 1, 2, 3, 4, 5	1

Name	Status	Category	Description	Values	Default Value
replace_sameid_rule_order	CHANGED	Assembly	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.	disable, 1, 2, 3, 4, 5	5
replace_samename_rule_order	CHANGED	Assembly	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.	disable, 1, 2, 3, 4, 5	2
replace_sameparam_rule_order	CHANGED	Assembly	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.	disable, 1, 2, 3, 4, 5	4
replace_unrelated_automatic	CHANGED	Assembly	Availability of automatic reference pairing in unrelated replace. No - Automatic reference pairing is not available. Yes - Automatic reference pairing is available.	yes, no	no
reserved_menu_space	CHANGED	User Interface	Reserves space for extra menus to the right of the Pro/ENGINEER window (1.0-4.0). This option overrides windows_scale.Reserves space for extra menus to the right of the Pro/ENGINEER window (0.0-4.0). This option overrides windows_scale.		1
round_displayed_dim_values	NEW	Dimensions & Tolerances	No - newly created or first shown dimension is displayed with all its significant decimals."Rounded Dimension Value" Check Box will be unchecked while opening Dimension Properties Dialog. Yes - number of decimals is rounded according to default dec places option."Rounded Dimension Value" Check Box will be checked while openingDimension Properties Dialog.)	yes, no	yes
save_drawing_picture_file	CHANGED	Drawing	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
saving_model_thumbnails	NEW	File Storage & Retrieval	Enables saving of model thumbnails.	yes, no	yes

Name	Status	Category	Description	Values	Default Value
shade_quality	NEW	Model Display	Model shade quality is used to decide how much to subdivide surfaces forshading purposes. Higher shade quality is slower but produces better pictures.	3, 1, 2, 4, 5, 6, 7, 8, 9, 10	3
show_annotations_dialog_limit	NEW	Drawing	Default value for Annotations dialog box.		
show_description_view_mgr	NEW	Without category	Yes - show a column for Description field for all tabs in view manager.	yes, no	no
sim_auto_map_max_dist	CHANGED	Mechanica	Sets the default maximum distance tolerance for auto-mapping of pressure load.Sets the default maximum distance tolerance (relative to model size) for auto-mapping of pressure load. The default value is 0.0012.		0.0012
sim_auto_pairing_thickness	NEW	Mechanica	Set the value of characteristic thickness for auto-detection of shell pairs. Using this as a threshold value, auto-detection will find surface pairs with thickness smaller than this value.		0
sim_check_jacobian_corners	NEW	Mechanica	No - Suppress Jacobian checks at corner nodes. ANSYS and Nastran checks parabolic elements Jacobian at the integration points only and FEM will fix too curved elements for the same criterion. Yes - Jacobian is checked both at integration points and corner nodes. FEM mode will fix too curved elements accordingly.	yes, no	no
sim_display_current_csys_triad	NEW	Without category	Toggles display of the current coordinate system triad in the lower right corner of the window.	yes, no	
sim_display_highlight_refs	NEW	Mechanica	Yes -Geometrical references are highlighted when selecting a load/constraint.	yes, no	
sim_display_load_name	NEW	Mechanica	Yes - The load/constraint name appears in the label.	yes, no	
sim_display_mesh_control_name	NEW	Mechanica	Yes - The Mesh/AutoGEM control name appears in the label.	yes, no	
sim_display_mesh_control_value	NEW	Mechanica	Yes - The Mesh/AutoGEM control value appears in the label.	yes, no	
sim_display_shell_pairs	NEW	Without category	Toggles the display of shell pairs.	no, yes	
sim_display_z_buffered	NEW	Without category	Toggles z-buffered display of Mechanica icons.	yes, no	
sim_ext_distribution_dir	NEW	Without category	Specifies the location of external distribution files for loads.		

Name	Status	Category	Description	Values	Default Value
sim_extend_surf_max_angle	NEW	Mechanica	Controls the maximum angle between a surface to be extended during shell compression and its adjacent surfaces. If the angle is higher than specified, the adjacent surfaces is extended. The default is 30 degrees. The range is from 0 to 90 degrees.		30
sim_icon_distribution_density	NEW	Mechanica	Determines how many icons are drawn for a distribution. 1 is fewer, 10 is the highest.		-1
sim_ignore_unpaired	NEW	Mechanica	Yes - Unpaired surfaces are ignored when meshing the model, placing you directly into the AFTER COMP menu (the CONT MESH menu is skipped).	yes, no	no
sim_pp_display_beam_thickness	NEW	Mechanica	Sets the thickness for beams and 2D-shells display in Post Processor. Numerical values specify the thickness in pixels.	actual, 1, 2, 3, 4, 5	5
sim_pp_path_absolute	CHANGED	Mechanica	Choose Absolute (Yes) or Relative (No) path to "rwd" file.	yes, no	no
sim_pp_show_cap_cut_outline	NEW	Mechanica	Enables to hide the model outline in capping/cutting surface display.	yes, no	yes
sim_remove_unopposed	NEW	Mechanica	Yes - The system ignores unopposed surfaces when meshing the model; the system also deselects the UseUnopposed option. No - the system uses the unopposed surfaces in the shell model.	yes, no	no
sim_run_copy_fem_neutral_file	DELETED	Mechanica	Controls whether the FEM neutral file (with extension .fnf) is copied into the study directory. The default value is YES.	yes, no	yes
simprep_default_model_status	NEW	Assembly	Determines the default model status while defining a new simplified rep.	exclude, master, graphics, geometry	exclude
simulation_product	CHANGED	Mechanica	Set Mechanica product being selected (Structure/Thermal). Prompt (default) -Brings Mechanica Model Type Dialog. This option is used only if the information about Mechanica product is not yet stored with model.	structure, thermal, prompt	prompt
sketcher_dec_places	DELETED	Sketcher	Sets the default number of decimal places displayed for dimensions in Sketcher. The extrusion depth or any other dimension you enter to create 3D geometry is controlled by the default_dec_places option.		2
sketcher_dim_of_revolve_axis	CHANGED	Sketcher	If this option is set all dimensions created by Intent Manager to Axis of Revolution will be diameter dimensions.	no, yes	yes

Name	Status	Category	Description	Values	Default Value
sketcher_intent_manager	DELETED	Sketcher	Specifies which sketcher to use. Yes - Intent Manager is the default. No - Old sketcher is the default.	yes, no	yes
sketcher_strngthn_to_def_d ec_pl	NEW	Without category	This option enable the user to keep a highly precise value during creation or convertion strong dimension.	yes, no	yes
smaller_bundle_min_bend_r ad	DELETED	Without category	Setup whether to consider the thickness of the bundle when set up the minimum bend radius.	yes, no	no
snap_single_constraints	NEW	Without category	When dynamically dragging components during component placement, allow to snap to references one constraint at a time.	yes, no	yes
startup_check_for_visualizati on	NEW	Environment	Determines whether or not the visualization components are checked for installation upon startup.	yes, no	yes
step_export_format	CHANGED	Data Exchange	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, ap214_cd, ap202_is, 203_is_ext, ap214_dis,	ap203_is
style_auto_regen_curves	NEW	Without category	Yes - Enable autoregeneration of curves.	yes, no	yes
style_auto_regen_shaded_s urface	NEW	Without category	Yes - Enable autoregeneration of shaded surfaces.	yes, no	yes
style_auto_regen_surface	NEW	Without category	Yes - Enable autoregeneration of surfaces.	yes, no	yes
style_connection_arrow_scal e	NEW	Without category	Set scale factor of surface connection arrow. Number should be greater then 0.		1
style_display_grid	NEW	Without category	Yes - Display grid for active datum plane.	yes, no	yes
style_grid_spacing	NEW	Without category	Set the number of lines to display on the grid. Number should be equal or greater than 2.		5
style_surface_mesh_display	NEW	Without category	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
style_surface_mesh_quality	NEW	Without category	Sets the quality of the surface mesh. Values range is between 0.1 to 10.		1
summary_help	CHANGED	User Interface	Refers to the one-line help Messages that are either displayed or not displayed.	yes, no	no
suppress_appearance_mess age	NEW	Colors	Suppress the lightworks appearance conversion notification dialog.	yes, no	no

Name	Status	Category	Description	Values	Default Value
template_mfgcast	CHANGED	File Storage & Retrieval	Specify the model used as the default manufacturing cast template.		inlbs_mfg_cast.asm
template_mfgcmm	CHANGED	File Storage & Retrieval	Specify the model used as the default manufacturing cmm template.		inlbs_mfg_cmm.asm
template_mfgemo	CHANGED	File Storage & Retrieval	Specify the model used as the default manufacturing expert machinist template.		inlbs_mfg_emo.asm
template_mfgmold	CHANGED	File Storage & Retrieval	Specify the model used as the default manufacturing mold template.		inlbs_mfg_mold.asm
template_mfgnc	CHANGED	File Storage & Retrieval	Specify the model used as the default manufacturing assembly template.		inlbs_mfg_nc.asm
tol_num_digits_default_driven	CHANGED	Drawing	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
ttf_handling_for_plot_files	NEW	Printing & Plotting	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
ui_theme	CHANGED	Without category	Setup Pro/E Theme.Standard will result in the Wildfire Theme on UNIX and "following the Windows theme on Windows XP & Win2K. Wildfire - Will always use the Wildfire Theme regardless of the OS settings.	wildfire_5, wildfire, standard	wildfire_5

Name	Status	Category	Description	Values	Default Value
units_system_dic_file	CHANGED	Piping	Specifies the file containing the mapping for MCAT filename and units.		<proe_WF_5.0_loadpoint>/text/piping_data/master_catalog/piping_units_system_dic_file.ptd
update_copy_geom_location	NEW	Assembly	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
use_cadam_plot_data	DELETED	Data Exchange	Determines whether the information in the Plot Axis System element should be taken into account when importing a CADAM drawing.	yes, no	no
use_outdated_accel_in_geom_rep	DELETED	File Storage & Retrieval	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.	no, yes	no
web_browser_homepage	CHANGED	System	Enter location of Pro/E browser home page.		<proe_WF_5.0_loadpoint>/html/chinese_tw/proe/helpsystop/resource_center_index.htm
web_enable_subscription	CHANGED	Application Programming Interfaces	Enable the subscription mechanism for Connections	no, yes	yes
weld_fillet_preferences_file	NEW	Without category	Sets the path and name of the fillet weld preferences file.		
weld_general_preferences_file	NEW	Without category	Sets the path and name of the general weld preferences file.		
weld_geom_type_default	CHANGED	Weld	Sets the default weld geometry type.	surface, light	surface
weld_groove_preferences_file	NEW	Without category	Sets the path and name of the groove weld preferences file.		
weld_plug_slot_preferences_file	NEW	Without category	Sets the path and name of the plug/slot weld preferences file.		
weld_spot_preferences_file	NEW	Without category	Sets the path and name of the spot weld preferences file.		

Name	Status	Category	Description	Values	Default Value
windows_browser_type	NEW	System	Select the type of embedded browser to use.	ie_in_process, ie_in_sep_process, mozilla_based_bro wser	
workpiece_instances	CHANGED	Without category	Cut out are created in Workpiece but not the family table instances.Instances are created in Workpiece for each steps.	steps, operations_and_st eps, none,	none