

Pro/ENGINEER[®]

Wildfire[™] 2.0

Configuration Options

Parametric Technology Corporation

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Pro/ENGINEER Wildfire 2.0 Configuration Options

Name	Category	Description	Values	Default Value
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_out_version	Without Category	Allows the user to choose ACIS export version.	public 4, public 5, internal 6, internal 7	5
activate_sketch_with_default	Features	Activate sketcher automatically using system defaults.	yes, no	no
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_components_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. No - System excludes welds when calculating mass properties.	yes, no	no
adobe_distiller	Printing & Plotting	Sets the Adobe Distiller command path. Required for PDF and COLOR PDF creation.	-	-
advanced_intersection	Assembly	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	no
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		yes, no	no

Name	Category	Description	Values	Default Value
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_float_opt_checkout	System	Yes - Allows checkout failure for floating modules.	no, yes	no
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_multiple_tool_tips	Manufacturing	Allows to use for machining cutting tool with multiple tips.	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
allow_ply_cross_section	Miscellaneous	Yes - Enables Pro/COMPOSITE to create a cross-section of composite plies.	yes, no	yes
allow_redo_intersections	Assembly	Yes - Displays the ReIntersect command in the ASSY FEAT menu.	yes, no	no
allow_ref_scope_change	Reference control	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

Name	Category	Description	Values	Default Value
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_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	no
allow_udf_style_cosmetic_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_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
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

Name	Category	Description	Values	Default Value
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
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

Name	Category	Description	Values	Default Value
assy_mfg_open_mode	Manufacturing	No Default Application set for opening Assembly NC Models.	internal none, public mfg, public process, public feature, internal toolmaker	-
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

Name	Category	Description	Values	Default Value
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_constr_always_use_offset	Assembly	Control whether auto constraint should create offsets. Yes - Auto constraint always creates offsets. No - Auto constraint snaps align or mate if surfaces are within tolerance. Never - Auto constraint never create offsets.	yes, no, never	no
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_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_traceback	System	When set to yes, traceback.log file will be generated when fatal error occurs.	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

Name	Category	Description	Values	Default Value
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
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.	-	-
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
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

Name	Category	Description	Values	Default Value
blank_layer	Layers	Blanks specified layers when you begin a Pro/ENGINEER session. The value is the layer ID.	-	0
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	public default, public excel_linked, public excel_embedded, internal generic_graphtool	default
bmgr_pref_file	User Interface	Specifies the location of the Graph Tool preference file.	-	-
bmx_param_restrictions	Without Category		yes, no	yes
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

Name	Category	Description	Values	Default Value
cable_int_portions_for_clear	Electromechanical	No - Global clearance checking for internal cable portions is excluded. Yes - Global clearance checking for internal cable portions is included.	no, yes	no
cadam_line_weights	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
cadds_import_layer	Data Exchange	Allows import of CADD55 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_refs	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
catia_translator	Data Exchange	Allows the user to choose between the old CATIA license-based converter (pro_cat) and the new non-CATIA license based (cat_ii) converters when doing direct import/export of CATIA files. When "topobus_enable" is set to "yes", cat_ii will be always used.	cat_ii, pro_cat	cat_ii
cdt_transfer_details	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

Name	Category	Description	Values	Default Value
centimeter_grid_interval	Features	Modify grid space in centimeter units for handle movement.	-	0.1
cgm_inc_pad_byte_in_length	Data Exchange	Yes - Enables a metafile to be processed by the Micrographic CGM converter.	yes, no	no
cgm_use_enum_in_real_spec	Data Exchange	Yes - Enables a metafile to be viewed in Advanced Technology Center's ForReview.	yes, no	no
cgm_use_reversed_ieee_floats	Data Exchange	Yes - Enables a metafile to be viewed in Advanced Technology Center's ForReview.	yes, no	yes
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_ghost_features	Without Category	Skip features that are already marked non-ghost.	skipchecked, never, always	skipchecked
check_same_assy_parts	Assembly	Determines whether to permit different names for different part occurrences, or not in mirror subassembly UI.	yes, no	yes
chk_part_surfs_profpack	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

Name	Category	Description	Values	Default Value
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
color	Colors	Turns the colors on or off. Turning colors off displays the model wireframe in white.	no, yes	yes
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
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	Constrain_in_window - New assembly components are shown in a separate window and must be constrained within the assembly. Package - New assembly components are shown within the assembly as a packaged component.	package, constrain_in_window	-

Name	Category	Description	Values	Default Value
comp_offset_dec_places	Assembly	Sets the default number of decimal places (0-10) to display in the offset column of component placement.	-	4
comp_rollback_on_redef	Assembly	No - Assembly is not rolled back when the user redefines a component.	yes, no	yes
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
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	You can compress object files to store them. Compressed files are slower to read and write, one-half to one-third the size, and fully compatible across systems. Yes - Stores object files in compressed format. No - Object files are not compressed.	yes, no	no
conf_mouse_anim	Miscellaneous	Display the presenter's mouse motions when observing a shared Pro/ENGINEER session.	no, yes	yes
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	no, yes	-

Name	Category	Description	Values	Default Value
		information.		
cpus_to_use	System	Set up the number of threads including main thread	-	2
create_drawing_dims_oly	Drawing	Yes - Saves all new dimensions created in the drawing (regardless of the drawing setup file option) inside the drawing as associative draft dimensions. No - Saves all dimensions created in Drawing mode in the part.	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
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	Miscellaneous	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	-

Name	Category	Description	Values	Default Value
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
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_assem_member, layer_assy_cut_feat, layer_axis, layer_chamfer_feat, layer_comp_design_model, layer_comp_fixture, layer_comp_workpiece, layer_copy_geom_feat, layer_corn_chamf_feat, layer_cosm_round_feat, layer_cosm_sketch, layer_csys, layer_curve, layer_curve_ent, layer_cut_feat, layer_datum, layer_datum_plane, layer_datum_point, layer_detail_item, layer_dgm_conn_comp, layer_dgm_highway, layer_dgm_rail, layer_dgm_wire, layer_dim, layer_draft_constr, layer_draft_dim, layer_draft_dtm, layer_draft_entity, layer_draft_feat, layer_draft_geom, layer_draft_grp, layer_draft_hidden, layer_draft_others, layer_draft_refdim, layer_driven_dim,	-

Name	Category	Description	Values	Default Value
			layer_dwg_table, layer_ext_copy_geom_feat, layer_feature, layer_geom_feat, layer_gtol, layer_hole_feat, layer_intchg_funct, layer_intchg_simp, layer_nogeom_feat, layer_nongraph_annot_elem, layer_note, layer_parameter_dim, layer_part_refdim, layer_point, layer_protrusion_feat, layer_quilt, layer_refdim, layer_ribbon_feat, layer_rib_feat, layer_round_feat, layer_sfin, layer_shell_feat, layer_skeleton_model, layer_slot_feat, layer_snap_line, layer_solid_geom, layer_surface, layer_symbol, layer_thread_feat, layer_trim_line_feat, layer_weld_feat	
default_abs_accuracy	Environment	Defines the default absolute part accuracy.	-	0
default_ang_dec_places	Drawing	Specifies the number of decimal places shown in angular dimensions in a drawing.	-	1
default_comp_geom_scope	Reference control	Set the specified Shared Geometry reference control conditions to newly added components.	all, pg, none	all

Name	Category	Description	Values	Default Value
default_comp_invalid_refs	Reference control	Set the specified handling method of out-of-scope Accessible references to newly added components.	copy, prohibit	copy
default_comp_scope_setting	Reference control	Set the specified Accessible reference control conditions to newly added components.	all, none, subassemblies, skeleton_model	all
default_comp_skel_geom_scope	Reference control	Set the specified Shared Geometry reference control conditions to newly added Skeleton components.	all, pg, none	all
default_comp_skel_scope_setting	Reference control	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 (0-14) 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.	-	2
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

Name	Category	Description	Values	Default Value
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_geom_scope	Reference control	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	Reference control	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	Reference control	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	Reference control	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	Reference control	Set default condition for reference control. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all, none, subassemblies, skeleton_model	all
default_placement_scope	Reference control	Default value for Component Placement Reference filter.	all, intrf_if_exists, intrf_only	all
default_ramp_size	Model Display	Specifies a number of shades of gray to show shading of non-colored objects.	-	0

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.	-	0
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_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
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_model_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 - Displays model with all lines in white. Hiddenviz -Hidden lines in grey. Hiddeninvis - All hidden lines removed. Shade - All models (except drawing) with all surfaces shaded. Line display is reflected in plotter, DXF and IGES files.	wireframe, hiddenviz, hiddeninvis, shade	shade
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_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

Name	Category	Description	Values	Default Value
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
display_layer	Layers	This causes the specified layers to display when you begin a Pro/ENGINEER session. The variable "value" is the layer Id.	-	0
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_silhouette_edges	Model Display	Sets the display of silhouette edges for wireframe display only.	yes, no	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.	public yes, internal no	no
display_tol_by_1000	Dimensions & Tolerances	If set to Yes, tolerances for non-angular dimensions will be displayed multiplied by 1000.	yes, no	no
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_mode	Data Management	Indicate which objects will be written to local cache when objects in Pro/E memory are saved.	none, modified, all	all

Name	Category	Description	Values	Default Value
dm_cache_size	Data Management	Enter amount of disk space (in megabytes) to allocate for local file storage.	-	400
dm_http_compression_level	Data Management	Set the data compression factor (0-9, 0= no compression) for data exchange with a Windchill server.	-	0
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
dm_remember_server	Data Management	Primary server/workspace must be set by user for each Pro/E session.	no, yes	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_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
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. Driven dimensions, gtols, and similar items cannot be added to the model. Cosmetic_only - Driven dimensions, gtols, cross sections, and explode states can be added to the model, design changes are not allowed.	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

Name	Category	Description	Values	Default Value
drawing_file_editor	Drawing	Specifies the default drawing text editor for the drawing setup file, otherwise, the system uses the default editor. Protab - the system uses Pro/TABLE. Editor - system uses the system editor (refer to pro_editor_command).	protab, editor	editor
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.	-	\$PTCSRC/text/prodetail.dtl
drawing_view_origin_systems	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	-
dwg_export_format	Data Exchange	Provides the ability to choose DWG file version when exporting from Pro/E drawing.	12, 13, 14, 2000	2000
dxblock_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
dx_export_format	Data Exchange	Provides the ability to choose DXF file version when exporting from Pro/E drawing.	12, 13, 14, 2000	2000
dx_export_mapping_file	Data Exchange	Specifies the DXF export mapping file. Absolute or relative path to the file can be used.	-	-
dx_in_faceted_brep	Data Exchange	Default, faceted boundary representation is allowed in import.	yes, no	yes
dx_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

Name	Category	Description	Values	Default Value
dxfl_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
dxfl_out_scale_views	Data Exchange	Specify whether or not to scale drawing views when exporting to the DXF or DWG file. Yes - rescales the entire drawing in such a way that the main view scale becomes 1:1. No - exports without rescaling.	yes, no	no
dxfl_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
dxfl_out_stroke_text	Data Exchange	Strokes == text is converted to lines and dots. Yes - Strokes out text in DXF or DWG export. No - Does not stroke out the text for export.	yes, no	no
dxfl_in_sjis	Data Exchange	Determines if the SJIS is supported when importing/exporting DXF files in Japanese	yes, no	no
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_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_placement_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	no
ecad_import_relative_accuracy	Data Exchange	Defines the relative accuracy used in ECAD import.	-	0.0012
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_other_outl_csyst_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_panel_csyst_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	Setup the default for the import of ECAD Holes from type PIN (IDF 3.0 ONLY)	yes, no	no
ecad_via_hole_import	Data Exchange	Setup the default for the import of ECAD Holes from type VIA (IDF 3.0 ONLY)	yes, no	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
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_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		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

Name	Category	Description	Values	Default Value
enable_k01_msr_anls	Without Category	Enable or disable new UI for Measure Analysis.	yes, no	no
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_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.	-	-
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_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 file names should be used for export to STEP (only). Yes - Do not prompt for a new name. No - Prompt for the name of each STEP file.	yes, no	no
export_to_shipit	Data Exchange	Enable Ship-it interface export.	yes, no	no

Name	Category	Description	Values	Default Value
extend_cvpath	Data Exchange	Extends CVPATH search path for CADD5 data exchanging.	-	-
fail_ref_copy_when_mising_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.	no, yes	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 redisplay more quickly at the expense of certain details. This option may increase memory usage slightly.	no, yes	no
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

Name	Category	Description	Values	Default Value
fem_allow_named_mesh_files	FEM	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_annotatations	FEM	Toggles transfer of loadcase names to ANSYS.	yes, no	no
fem_ansys_grouping	FEM	Toggles transfer of groups/layers to ANSYS.	yes, no	no
fem_asp_ratio	FEM	The value that FEM created elements of aspect ratio are compared against.	-	7
fem_default_solver	FEM	Specifies default solver.	ansys, nastran	-
fem_dist_index	FEM	The value that FEM created elements of distortion indices are compared against.	-	0.4
fem_edge_angle	FEM	Sets the minimum acceptable default angle between adjacent element edges. Enter value between 0 and 90.	-	30
fem_ignore_unpaired	FEM	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
fem_mesh_operations	FEM	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	FEM	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	FEM	The value that FEM created elements of mid ratios are compared against.	-	0.1
fem_neutral_version	FEM	Specifies version number of neutral file.	-	3

Name	Category	Description	Values	Default Value
fem_remove_unopposed	FEM	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
fem_skew_angle	FEM	Sets the maximum acceptable default skew angle value, measured in degrees. Enter a value between 0 and 90.	-	45
fem_solid_shell_auto_constraint	FEM	Yes - SPCs for the rotational DOF will automatically be created at the interfacing nodes between solids and shells. That is shell rotations will be fixed. No - No such constraints will automatically be created. Ask - Prompt the user before running.	yes, no, ask	no
fem_solver_time_limit	FEM	For NASTRAN solution, uses this option to interrupt the solver after the specified time limit. The default value is 60 minutes.	-	60
fem_taper	FEM	Sets the minimum acceptable default taper value. Enter a value between 0 and 1.	-	0.5
fem_twist_angle	FEM	Sets the maximum acceptable default twist angle between opposing element faces. Applies to wedge and brick elements only. Enter a value between 0 and 90.	-	10
fem_warp_angle	FEM	Sets the maximum acceptable default warp angle value, measured in degrees. Enter a value between 0 and 90.	-	10
fem_which_ansys_solver	FEM	Specifies the ANSYS solver to be used.	frontal, iterative, powersolver	frontal
fem_y_dir_compress	FEM	Specifies the Y-direction based on compressed and uncompressed geometry. The default value is uncompressed.	yes, no	no
file_open_default_folder	File Storage & Retrieval	Sets default dir for File>Open. Working_directory - Search working directory. In_session - Search objects in session. Pro_library - Use only if Pro/LIBRARY is installed. Workspace - Use only with Pro/INTRALINK. Commonsplace - Use only with	default, working_directory, in_session, pro_library, my_documents	-

Name	Category	Description	Values	Default Value
		Pro/INTRALINK.		
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_boundaries_on_import	Data Exchange	Controls fixing of boundaries of imported surfaces. Yes - Fixes boundaries by zipping gaps and correcting tangencies. No - Does not fix boundaries.	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
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
folder_display_file_limit	File Storage & Retrieval	Sets the default number of files that will be displayed when viewing the contents of a folder using the Folder Navigator.	-	-1
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

Name	Category	Description	Values	Default Value
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
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
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
gerber_360_interpolation	Printing & Plotting	Yes - Arcs and circles included in outputted Gerber plot files will take this format.	no, yes	no
gpostpp_dir	Manufacturing	Specifies the directory containing the post processor files to be used by Pro/NCPOST (GPOST).	-	-

Name	Category	Description	Values	Default Value
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.	public opengl, public x_windows, public starbase, public xgl, public win32_gdi, internal no_graphics	no_graphics
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
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_tang_line_display	Electromechanical	Display the tangency lines between cable segments in thick cable display mode.	yes, no	yes
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_layer_items	Layers	Specifies whether items selected in the layer tree should be highlighted in the graphics area.	yes, no	yes
highlight_new_dims	Drawing	In Drawing mode, highlights new dimensions in red until you move them or repaint the screen.	yes, no	no
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

Name	Category	Description	Values	Default Value
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_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
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_106_f2_as_spline	Data Exchange	No - IGES Copious Data Entity type 106, form 2 is imported as a set of points in 3D space. Yes - The entity is imported as a 3D spline.	no, yes	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 geom info of comps and assy feats.	public flat, internal one level, internal all levels, public one_level, public all_levels, internal all parts, public 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

Name	Category	Description	Values	Default Value
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
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_srf_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

Name	Category	Description	Values	Default Value
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_srf_s_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	Reference control	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
ignore_export_geom_declaration	Reference control	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.	public text, public html, internal xml_text, internal dbg_text	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

Name	Category	Description	Values	Default Value
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
instance_search_exhaustive	File Storage & Retrieval	Specifies extent of instance search. Yes - Include file type when entering instance name in File>Open: type in input panel (e.g. inst_name.prt) or select in drop-down list. If no .idx files are found, only working dir is searched; each generic is not.	no, yes	no
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 CADDSS5 model.	yes, no	no
intf_cadds_version	Data Exchange	Allows the user to switch between the default and alternate converter version for interface with CADDSS5	internal 10, internal 11, public 12, public 13	-
intf_in_arclength_reparam	Data Exchange	Yes - During import, the system attempts to change parameters for every non-analytical surface (Tabulated Cylinders, Surfaces of Revolution, Ruled Surfaces, Spline Surfaces, B-spline Surfaces), whose parameters differ significantly from arc-length.	yes, no	no
intf_in_blanked_entities	Data Exchange	Filters the import of entities based on their blank status in the import file.	yes, no	yes
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

Name	Category	Description	Values	Default Value
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_treat_polyline_as	Data Exchange	Enables you to choose the representation of imported independent XYZ polylines. The system interprets the imported polylines based on which option is chosen and displays them accordingly.	public single_polyline, public single_spline, public set_of_curves, internal set_of_points	single_polyline
intf_in_use_template_models	Data Exchange	Allows the user to use template models in Pro/INTERFACE import.	yes, no	no
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_layer	Data Exchange	Layer mapping mechanism providing an ability to group entities to layers or blocks.	none, part_layer, block_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_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_use_variable_size	Data Exchange	Yes - IGES,DXF,STEP files with variable sized sheet information are imported and put on the appropriate variable-size format. Without a sheet size, the system tries to apply the correct variable-size format. No - The IGES,DXF,STEP drawing is placed on a standard sized format.	no, yes	yes
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

Name	Category	Description	Values	Default Value
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_iges_hatch_boundary_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_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_text_alignment	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_cgm_ver	Data Exchange	Provides the ability to choose CGM metafile version when exporting from Pro/E drawing.	1, 3	1
intf2d_out_enhanced_entities	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_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
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.	-	-

Name	Category	Description	Values	Default Value
intf3d_ideas_run_command	Data Exchange	Specifies the I-DEAS run command. Default command is "ideas".	-	ideas.cmd
intf3d_in_close_open_boundaries	Data Exchange	Controls fixing of open boundaries on imported surfaces.	yes, no	no
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_include_items	Data Exchange	Specifies what entity types you want to import from a file.	srfs_crvs_pnts, srfs_crvs, crvs_pnts, srfs_pnts, srfs, crvs, pnts	srfs_crvs_pnts
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_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

Name	Category	Description	Values	Default Value
intf3d_out_extend_surface	Data Exchange	Specifies how surfaces are handled for exporting files to other systems	yes, no	yes
intf3d_out_force_surface_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_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_parasolid_export_schema	Without Category		-	sch_10004
intf3d_ug_install_dir	Without Category	Specifies the path for Unigraphics installation. Use the full pathname to avoid problems.	-	-
isogen_attribute_map_file	Piping	Specifies the mapping between isogen parameter and attribute name.	-	\$PTCSRC/text/piping_data/isodata/isogen_attribute_map.ptd
isogen_endtype_map_file	Piping	Specify name of file mapping Pro/Piping endtypes with ISOGEN endtypes	-	\$PTCSRC/text/piping_data/isodata/isogen_endtype_map.ptd
isogen_mandatory_attribute_file	Piping	Specify file containing mandatory attribute filenames used by ISOGEN.	-	isogen.flc
isogen_nominal_size_map_file	Piping	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches	-	\$PTCSRC/text/piping_data/isodata/isogen_nominal_size_map.ptd
isogen_output_files_dir	Piping	Specify directory to save the PCFs created by ISOGEN interface	-	-

Name	Category	Description	Values	Default Value
isogen_pcf_filename_for_mat	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	-	\$PTCSRC/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
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
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

Name	Category	Description	Values	Default Value
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
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

Name	Category	Description	Values	Default Value
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
log_dwg_spr1150166_detection	Without Category	Create a log file and include if a particular drawing needs correction for SPR #1150166.	no, yes	no
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_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
mapkey	Miscellaneous	Stores a keyboard macro. Use Tools>Mapkeys to create, edit and delete mapkeys.	-	-
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
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

Name	Category	Description	Values	Default Value
maximum_texture_size	Model Display	Allocates a limit to texture size and controls the resizing of texture maps that are used inside Pro/ENGINEER graphics only. It has no effect on PhotoRender.	64x64, 128x128, 256x256, 512x512	-
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_density	Mechanism	Mass density for dynamics.	-	0
mdo_enable_joint_friction	Mechanism	Enable joint axis friction.	yes, no	no
mdo_explicit_integration	Mechanism	Enable explicit integration for dynamic analyses.	yes, no	no
mdo_spring_wire_radius	Mechanism	Spring wire radius for dynamics spring icons.	-	0.15
mdo_use_mm_body_order	Mechanism	Use Mechanica Motion body order. (Make the ground to be the first body in connection).	yes, no	no
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.	-	-

Name	Category	Description	Values	Default Value
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
merge_smt_srf_s_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
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_prt_as_check_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		-	-0.001
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_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.	-	-

Name	Category	Description	Values	Default Value
mfg_param_mode	Manufacturing	Simple - The parameter tree is shown with a subset of the parameters. Advanced - The parameter tree is shown with the entire set of parameters.	simplified, advanced	simplified
mfg_process_print_dir	Without Category	Specifies the directory where Mfg Process html page get stored. Use the full path name 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_start_model_dir	File Storage & Retrieval	Provides the complete path to the directory containing mfg start assemblies.	-	-
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_workpiece_transparency	Manufacturing		-	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
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

Name	Category	Description	Values	Default Value
model_allow_ref_scope_change	Reference control	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.	-	0
model_grid_spacing	Environment	Sets the default model grid spacing.	-	1.5
model_note_display	Environment	Allows users to display all shown model notes. Applies to shown notes, and does not change Show/Erased status of the notes.	yes, no	yes
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_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_enabled	Environment	Enable ModelCHECK	yes, no	yes
mold_layout_origin_name	Casting & Mold Design	Sets a specified coordinate system as the default for the cavity layout origin.	-	-

Name	Category	Description	Values	Default Value
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	Simulation	Displays the Motion Envelope alert the first time the quality level is increased.	yes, no	yes
mp_calc_level	Without Category	Specifies the level of mass properties calculation.	assembly_only, all_models	assembly_only
msg_translation	User Interface	Specifies language in which Messages are displayed when running non-English version of Pro/ENGINEER.	yes, no	yes
multiple_skeletons_allowed	Without Category	Allows/restricts multiple skeleton.	yes, no	no
multipoint_location_count	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
native_kbd_macros	User Interface	Specifies support of keyboard macros written in native language (for example, German).	no, yes	no
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.	-	\$PTCSRC/text/ncmdl_data/ncmdl_bar.prt

Name	Category	Description	Values	Default Value
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.	-	\$PTCSRC/text/ncmdl_data/ncmdl_billet.prt
ncpost_type	Manufacturing	Set to GPOST to access the default post processor supplied with Pro/NCPOST.	gpost, ncpost	-
new_asm_regen_revnums	Assembly	Yes - Regenerate command will not increase the model revision number of the top assembly unless features belonging to the top assembly actually get regenerated. Regenerate command will increase model revision numbers of subassemblies if features in the subassembly regenerate. No - Regenerate command will always increase the revision number of the top assembly and almost never increase the revision numbers of subassemblies, regardless of whether or not assembly features regenerate.	yes, no	yes
new_designate_ui	Without Category		yes, no	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
nt_cache_dirs	Without Category	Addresses performance issues that occur when Pro/ENGINEER is used with NTFS-mounted file systems. See Technical Support TAN for more information.	yes, no	-
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

Name	Category	Description	Values	Default Value
offset_line_def_font	Assembly Process	Specifies the font in which offset lines are displayed in drawings.	-	-
open_simplified_rep_by_default	File Storage & Retrieval	Name of rep to retrieve upon file open or "Yes" to always invoke open rep dialog.	-	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	yes
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

Name	Category	Description	Values	Default Value
param_dec_places	User Interface	Sets the number of decimal places (0-14) to display in parameters.	-	6
parenthesize_ref_dim	Dimensions & Tolerances	Encloses reference dimensions in parentheses. No - Follows reference dimensions with the text "REF".	yes, no	no
pdf_plot_config	Printing & Plotting	Set the pcf filename to be used when saving to PDF format.	-	-
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
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
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

Name	Category	Description	Values	Default Value
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_csyes_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_convert	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.	-	\$PTCSRC/text/piping_data/pipeline_library
pipeline_assembly_name_format	Piping	Sets the default pipeline assembly name format.	-	mnemonic,- ,number

Name	Category	Description	Values	Default Value
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]spoolnu m[01]suffix[[]
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.	-	\$PTCSRC/text/pi ping_data/master _catalog/piping_a ppearance.map
piping_bolt_nut_selection_file	Piping	Piping bolts and nuts selection file.	-	\$PTCSRC/text/pi ping_data/master _catalog/piping_b olt_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_end_compatibility_file	Piping	Sets the default piping end compatibility filename.	-	\$PTCSRC/text/pi ping_data/master _catalog/piping_e nd_compatibility. ptd
piping_extend_dimension_scheme	Piping	Specify a dimension scheme for piping extend segment.	cartesian, cylindrical, spherical	spherical
piping_fitting_angle_tolerance	Piping	Sets the default piping fitting angle tolerance value.	-	1.5

Name	Category	Description	Values	Default Value
pipng_fitt_category_map_file	Piping	Sets the default piping fitting category map filename.	-	\$PTCSRC/text/piping_data/sample_project/piping_fitt_category_map.ptd
pipng_fitt_library_dir	Piping	Sets the default piping fitting library directory.	-	\$PTCSRC/text/piping_data/fittinglib
pipng_fluid_parameter	Piping	Enables/disables the Parameters tab in Create Pipeline Dialog	no, yes	no
pipng_insulation_dir_file	Piping	Sets the default piping insulation directory filename.	-	\$PTCSRC/text/piping_data/sample_project/piping_insulation_dir.ptd
pipng_joint_fitting_clearance	Piping	Sets the piping joint minimum fitting clearance value.	-	1
pipng_joint_fitting_offset	Piping	Sets the piping joint minimum fitting offset value.	-	1
pipng_manufacture_dir_file	Piping	Sets the default piping manufacture directory filename.	-	\$PTCSRC/text/piping_data/master_catalog/piping_manufacture_dir.ptd
pipng_material_file	Piping	Sets the default piping material filename.	-	\$PTCSRC/text/piping_data/master_catalog/piping_material.ptd
pipng_mcat_dir	Piping	Sets the default piping master catalog directory.	-	\$PTCSRC/text/piping_data/master_catalog
pipng_mcat_dir_file	Piping	Sets the default piping master catalog directory filename.	-	\$PTCSRC/text/piping_data/master_catalog/piping_mcat_dir.ptd

Name	Category	Description	Values	Default Value
pipng_min_bolting_clearance	Piping	Sets the default piping minimum bolting clearance value.	-	1
pipng_project_data_dir	Piping	Sets the default Project data directory.	-	\$PTCSRC/text/piping_data/sample_project
pipng_schematic_driven	Piping	Set the Schematic Driven Pipeline Modeling ON/OFF	no, yes	no
pipng_schematic_xml_dir	Piping	Specify directory with full path containing schematic information XML file(s).	-	c:/tmp/dump
pipng_spec_dir_file	Piping	Sets the default piping specification directory filename.	-	\$PTCSRC/text/piping_data/sample_project/piping_spec_dir.ptd
pipng_system_tree_format	Piping	Sets the default piping model tree format.	-	mnemonic
pipng_thumb_wheel_increment	Piping	Sets the default piping thumb wheel increment value.	-	1
pipng_thumb_wheel_ratio_inc	Piping	Sets the default piping thumb wheel ratio increment value.	-	0.01
pipng_wildfire_convert	Piping	Always ask before converting old piping assemblies to update the network structure.	manual, never, automatic	manual
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 (except PostScript plots) are simply given the extension .plt, as before.	yes, no	-

Name	Category	Description	Values	Default Value
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	-
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_save_as_type	File Storage & Retrieval	Allows customizing the order of the Type selections found under File ->Save A Copy.	-	-

Name	Category	Description	Values	Default Value
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.	internal none, internal all, internal geom_only, public yes, public 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.	-	-
pro_ansys_path	FEM	Specifies path to ANSYS executable.	-	-
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.	-	-

Name	Category	Description	Values	Default Value
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 drawing_file_editor and/or part_table_editor and/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).	-	\$PTCSRC/text
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.	-	\$PTCSRC/i486_n t/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.	-	-
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.	-	-

Name	Category	Description	Values	Default Value
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_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	FEM	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.	-	c:/tmp/dump
pro_palette_dir	Drawing	Sets the default directory for drawing symbol palette files.	-	\$PTCSRC/symbols/palette
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.	-	-

Name	Category	Description	Values	Default Value
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	FEM	Specifies the name of the user-defined solver to be included in the SOLVE menu. See also pro_solver_path.	-	-
pro_solver_path	FEM	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.	-	c:/tmp/dump
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.	-	\$PTCSRC/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.	-	c:/tmp/dump
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
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

Name	Category	Description	Values	Default Value
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.	-	-
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_display	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
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

Name	Category	Description	Values	Default Value
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
ptncpost_dbf	Manufacturing	Enter directory path to NCPPost database file.	-	-
punch_axis_points	Sheetmetal	Yes - Enables the creation of punch axis points in Sheetmetal cuts and punches.	yes, no	no
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
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

Name	Category	Description	Values	Default Value
read_vda_in_pset_as_spline	Data Exchange	Specifies whether a VDA Pset entity is imported to Pro/ENGINEER as a spline.	yes, no	yes
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	Reference control	Set color of backedup references.	-	55
ref_scope_copy_color_change	Reference control	Enable changing colors of backedup references during selection.	no, yes	yes
ref_scope_no_pick_to_copy	Reference control	Disable selecting references backedup by references control.	no, yes	no
ref_scope_no_pick_to_prohibit	Reference control	Disable selecting references prohibited by references control.	no, yes	yes
ref_scope_prohibit_color	Reference control	Set color of prohibited references.	-	45
ref_scope_prohibit_color_change	Reference control	Enable changing colors of prohibited references during selection.	no, yes	yes
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_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
regenerate_read_only_objects	Data Management	Determines whether an object fetched from a Pro/INTRALINK database with blocking 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

Name	Category	Description	Values	Default Value
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
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
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
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
reserved_menu_space	User Interface	Reserves space for extra menus to the right of the Pro/ENGINEER window (1.0-4.0). 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.	-	-

Name	Category	Description	Values	Default Value
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_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.	-	0
right_mouse_button_pop_up	User Interface	Specify whether or not the right mouse button pop up menu is available or not.	yes, no	yes
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	-
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_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_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	no

Name	Category	Description	Values	Default Value
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 how instances are saved. None - Instance accelerator files not used. Explicit - Save instance accelerator files when instances explicitly saved. Always - Always save instance accelerator files.	none, explicit, always	none
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 View Performance dialog (shaded).	public wireframe, public shading_low, public shading_high, public shading_lod, internal no_display	shading_lod
save_modified_drawings_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
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

Name	Category	Description	Values	Default Value
scope_invalid_refs	Reference control	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_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.	-	-
section_color	Sketcher	Specifies the color of sketched sections.	default, drawing_color	-
sel_insts_on_comp_retrieval	Assembly	Yes - If the components used in the assembly instances are themselves generics, the system 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

Name	Category	Description	Values	Default Value
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
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_axes_for_extrudes	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

Name	Category	Description	Values	Default Value
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_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
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
silhouettes	Model Display	Specifies whether silhouette edges are displayed or not by default.	no, yes	yes
sim_additive_mass	Simulation	Treats Mass Idealizations as additive if they are applied to the same entity and suppresses Mass precedence rules.	yes, no	no
sim_asm_modeling	Simulation	Controls automatic creation of assembly links and free connections in Mechanics. The default is YES.	yes, no	yes
sim_beamsection_path	FEM	Specifies the path where the Beam Sections will be stored.	-	-
sim_display_agem_controls	Simulation	Toggles display of AutoGEM Controls.	yes, no	yes

Name	Category	Description	Values	Default Value
sim_display_arrow_scale	Simulation	Toggles automatic arrow scaling.	yes, no	yes
sim_display_arrow_tail_touching	Simulation	Toggles display of Load arrows to have tails or heads touching.	yes, no	no
sim_display_beam_releases	Simulation	Toggles display of Beam Release icons.	yes, no	yes
sim_display_beam_sections	Simulation	Toggles display of beam section icons.	yes, no	yes
sim_display_beams	Simulation	Toggles display of beam icons.	no, yes	yes
sim_display_contact_regions	Simulation	Toggles display of Contact Region icons.	yes, no	-
sim_display_fasteners	Simulation	Toggles display of Fastener icons.	yes, no	-
sim_display_gaps	Simulation	Toggles display of gap icons.	no, yes	-
sim_display_in_spin	Simulation	Toggles display of simulation objects in Dynamic Spin.	yes, no	yes
sim_display_interfaces	Without Category	Toggles display of structure connection icons.	no, yes	yes
sim_display_load_colors	Simulation	Toggles display of Load colors.	yes, no	yes
sim_display_load_distribution	Simulation	Toggles display of distributed load vectors over entire entity.	yes, no	yes
sim_display_load_icons	Simulation	Toggles display of load icons.	yes, no	yes
sim_display_load_value	Simulation	Toggles display of Load values.	yes, no	yes
sim_display_masses	Simulation	Toggles display of mass icons.	no, yes	-
sim_display_matl_assignments	Simulation	Toggles display of Material Assignment icons.	yes, no	yes
sim_display_measures	Simulation	Toggles display of Simulation Measure icons.	yes, no	yes
sim_display_mesh_and_model	Simulation	Toggles display of Mesh together with Model.	yes, no	no
sim_display_mesh_controls	Simulation	Toggles display of Mesh Controls.	yes, no	yes
sim_display_mesh_entities	Simulation	Toggles display of Mesh entities.	yes, no	yes

Name	Category	Description	Values	Default Value
sim_display_mesh_mode	Simulation	Specifies the mode of mesh display in FEM preprocessor. Values: NO_MESH, WIREFRAME, HIDDEN, NOHIDDEN, SHADING.	no_mesh, wireframe, hidden, nohidden, shading	wireframe
sim_display_mesh_quality	Simulation	Specifies the quality of mesh display in Mechanica. Values: FINE, MEDIUM, COARSE.	fine, medium, coarse	coarse
sim_display_mesh_shells_thick	Simulation	Toggles the display of shell thickness in Mechanica mesh. YES - Display shells with real thickness. NO - Display shells with zero thickness.	no, yes	no
sim_display_mesh_shrink_elems	Simulation	Controls display of Mesh through a specified percentage of shrinking of its elements.	-	-1
sim_display_motion_cams	Simulation	Toggles display of motion cam icons.	yes, no	yes
sim_display_motion_connections	Simulation	Toggles display of motion connection icons.	yes, no	yes
sim_display_motion_contact_rgns	Simulation	Toggles display of motion contact region icons.	yes, no	yes
sim_display_motion_design_vars	Simulation	Toggles display of design variable icons.	yes, no	yes
sim_display_motion_drivers	Simulation	Toggles display of motion driver icons.	yes, no	yes
sim_display_motion_gears	Simulation	Toggles display of motion gear icons.	yes, no	yes
sim_display_motion_ground_pnts	Simulation	Toggles display of motion ground point icons.	yes, no	yes
sim_display_motion_local_sys	Simulation	Toggles display of local coordinate systems icons.	yes, no	yes
sim_display_motion_loads	Simulation	Toggles display of motion load icons.	yes, no	yes
sim_display_motion_slots	Simulation	Toggles display of motion slot icons.	yes, no	yes

Name	Category	Description	Values	Default Value
sim_display_names	Simulation	Toggles display of Simulation Object names.	yes, no	no
sim_display_rigid_connections	Simulation	Toggles display of rigid connection icons.	yes, no	-
sim_display_rigid_dof	Simulation	Toggles display of Rigid Link DOF icons.	yes, no	-
sim_display_rigid_links	Simulation	Toggles display of Rigid Link icons.	yes, no	-
sim_display_shells	Simulation	Toggles display of shell icons.	no, yes	yes
sim_display_spot_welds	Simulation	Toggles display of spot weld icons.	yes, no	yes
sim_display_springs	Simulation	Toggles display of spring icons.	no, yes	-
sim_display_struct_constraints	Simulation	Toggles display of structure constraint icons.	yes, no	-
sim_display_struct_loads	Simulation	Toggles display of structure load icons.	yes, no	-
sim_display_therm_bcs	Simulation	Toggles display of Thermal Boundary Condition icons.	yes, no	-
sim_display_therm_loads	Simulation	Toggles display of thermal load icons.	yes, no	-
sim_display_weighted_dof	Simulation	Toggles display of Weighted Link DOF icons.	yes, no	-
sim_display_weighted_links	Simulation	Toggles display of Weighted Link icons.	yes, no	-
sim_display_welds	Simulation	Toggles display of Weld icons.	yes, no	yes
sim_fatigue_biaxiality_correct	Simulation	Controls the use of biaxiality correction. The default value is YES.	yes, no	yes
sim_fatigue_biaxiality_method	Simulation	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	Simulation	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_matedata	Simulation	Controls the use of external material data. The default value is NO.	no, yes	no

Name	Category	Description	Values	Default Value
sim_fatigue_hysteresis_gate	Simulation	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	Simulation	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	Simulation	Controls the application of the mean stress correction. The default value is YES.	yes, no	yes
sim_fatigue_mean_stress_method	Simulation	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
sim_fatigue_safety_margin	Simulation	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	Simulation	Specifies the directory for user files (for example, external material files). The default value is the current directory.	-	-
sim_fatigue_write_surface_stress	Simulation	Controls writing of surface stresses to a neutral file. The default value is NO.	no, yes	no
sim_fem_nastran_use_pshell	Simulation	Yes - use PSHELL cards, No - use PCOMP cards for laminate composite shells in NASTRAN output file.	yes, no	no
sim_html_report_preferences_file	FEM	Specifies the path to the Simulation post processor HTML report preferences file.	-	-
sim_load_mech_mesh	Simulation	No - The user is prompted to load the existing Mechanics mesh file from the current working directory. Yes - The mesh file, if exists, is loaded automatically and the mesh is displayed.	no, yes	yes

Name	Category	Description	Values	Default Value
sim_mat_poissons_notation	Simulation	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_memory_usage	Simulation	Specifies the amount of memory in megabytes to be used in Mechanica run and FEM mesher. The default value is 128.	-	128
sim_motion_analysis_accuracy	Simulation	Specifies the default accuracy to which Mechanica determines values at each increment of the analysis. This number represents the acceptable fractional error in the computation of joint axis positions and rates at each time increment of the analysis. The default value is 0.0001.	-	0.0001
sim_motion_analysis_assem_tol	Simulation	Specifies the default error allowed for the assembly analysis. The default value is 0.001.	-	0.001
sim_motion_analysis_duration	Simulation	Specifies the default length of time in seconds during which the analysis runs. The analysis duration can be less than the duration of the motion simulation. The default value is 10.0.	-	10
sim_motion_analysis_increment	Simulation	Specifies the default increment between steps at which Mechanica reports the motion of the mechanism. The default value is 0.1	-	0.1
sim_motion_analysis_integrator	Simulation	Controls the default integrator Mechanica uses when running the motion analysis. The value types are IMPLICIT (default) and EXPLICIT.	implicit, explicit	implicit
sim_motion_analysis_method	Simulation	Controls the default method Mechanica uses to analyze the motion of your model. The default is FASTEST_AVAILABLE.	fastest_available, time_integration, kinetostatic	fastest_available
sim_motion_analysis_start_time	Simulation	Specifies the default time in seconds at which the motion analysis begins. The default value is 0.	-	0

Name	Category	Description	Values	Default Value
sim_motion_analysis_velocity_tol	Simulation	Specifies the default error allowed for the velocity analysis. The default value is 0.001.	-	0.001
sim_motion_output_fly_file	Simulation	Controls the creation of .fra file when saving results for a motion model, The .fra file is used to import a motion animation into Pro/FLY-THROUGH. The default is NO.	no, yes	no
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	Simulation	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	Simulation	Specifies the background color of the work area for the results display. The default value is PROE.	proe, blue, black, white	proe
sim_pp_path_absolute	Simulation	Choose Absolute (Yes) or Relative (No) path to "rwd" file.	yes, no	yes
sim_pp_template_dir	Simulation	Sets the default directory for results template files.	-	-
sim_pp_vector_plot_arrow_2d	Simulation	"Yes" plots 2D arrows, "NO" plots 3D arrows in Vector Plot of Mechanics Postprocessor.	yes, no	no
sim_pp_vrml_export_for_mat	Simulation	Flags to denote VRML1.0 or VRML2.0 (default) output.	vrml2.0, vrml1.0	vrml2.0
sim_pp_vrml_feature_edges	Simulation	Flag to enable the inclusion of feature edges in the VRML file. The default value is NO.	no, yes	no

Name	Category	Description	Values	Default Value
sim_regen_on_entry	Simulation	Yes - Regenerate model on entrance to Simulation Application. No - Don't regenerate the model.	yes, no	yes
sim_run_copy_fem_neutral_file	Simulation	Controls whether the FEM neutral file (with extension .fnf) is copied into the study directory. The default value is YES.	yes, no	yes
sim_run_out_dir	Simulation	Specifies the directory for Mechanica run output storage. By default, the current directory is used.	-	-
sim_run_tmp_dir	Simulation	Specifies the directory for Mechanica run temporary storage. By default, the current directory is used.	-	-
sim_smooth_aspect_ratio	Simulation	Sets the element aspect ratio criterion for the mesh smoothing process.	-	7
sim_smooth_edge_angle	Simulation	Sets the element edge angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	30
sim_smooth_skew	Simulation	Sets the element skew angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	45
sim_smooth_taper	Simulation	Sets the element taper criterion (Quad elements only) for the mesh smoothing process.	-	0.5
sim_smooth_warp_angle	Simulation	Sets the element warp angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	10
sim_use_layers	Simulation	Allows layers to become groups for Mechanica Independent mode.	no, yes	no

Name	Category	Description	Values	Default Value
simulation_fem_mode	Simulation	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	Simulation	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	Simulation	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	Simulation	Specifies the default location of the Process Guide template files.	-	-
simulation_product	Simulation	Set Mechanica product being selected (Structure/Thermal/Motion). Prompt (default) - Brings Mechanica Model Type Dialog. This option is used only if information about Mechanica product is not yet stored with model.	public structure, public thermal, internal mold, internal motion, public 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
sketcher_animated_modify	Sketcher	No - The section is not animated as modifications are regenerated.	yes, no	yes
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_skamps	Sketcher	Yes - collinear constraints will be used by Intent Manager; No - collinear constraints will not be used by Intent Manager.	no, yes	yes

Name	Category	Description	Values	Default Value
sketcher_dec_places	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_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_skamps	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_skamps	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
sketcher_import_exact_geometry	Sketcher	During import in sketcher use exact geometry.	yes, no	yes
sketcher_intent_manager	Sketcher	Specifies which sketcher to use. Yes - Intent Manager is the default. No - Old sketcher is the default.	yes, no	yes

Name	Category	Description	Values	Default Value
sketcher_lineup_hor_skamps	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_skamps	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_skamps	Sketcher	Yes - midpoint constraints will be used by Intent Manager; No - midpoint constraints will not be used by Intent Manager.)	no, yes	yes
sketcher_num_digits	Sketcher	Enter a number of digits for scetcher accuracy.	-	2
sketcher_parallel_skamps	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_skamps	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_entry_skamps	Sketcher	Yes - point on entry constraints will be used by Intent Manager; No - point on entry 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

Name	Category	Description	Values	Default Value
sketcher_same_point_skamps	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 a x-grid spacing value to override the default x-grid spacing value.	-	1
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_symmetric_skamps	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_skamps	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_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

Name	Category	Description	Values	Default Value
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_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
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_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
spherical_map_size	Drawing	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

Name	Category	Description	Values	Default Value
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.	-	-
std_nastran_template	FEM	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

Name	Category	Description	Values	Default Value
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.	public ap203_is, internal ap203_dis, public ap214_cd, public ap202_is, public 203_is_ext, public ap214_dis, public ap209_dis, public ap214_is	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_default_connections	Without Category	Yes - Add connections to a newly created Style surface.	yes, no	yes
summary_help	User Interface	Refers to the one-line help Messages that are either displayed or not displayed.	yes, no	yes
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

Name	Category	Description	Values	Default Value
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
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

Name	Category	Description	Values	Default Value
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
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

Name	Category	Description	Values	Default Value
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
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_mfgcast	File Storage & Retrieval	Specify the model used as the default manufacturing cast template.	-	inlbs_mfg_cast.mfg
template_mfgcmm	File Storage & Retrieval	Specify the model used as the default manufacturing cmm template.	-	inlbs_mfg_cmm.mfg
template_mfgemo	File Storage & Retrieval	Specify the model used as the default manufacturing expert machinist template.	-	inlbs_mfg_emo.mfg

Name	Category	Description	Values	Default Value
template_mfgmold	File Storage & Retrieval	Specify the model used as the default manufacturing mold template.	-	inlbs_mfg_mold.mfg
template_mfgnc	File Storage & Retrieval	Specify the model used as the default manufacturing assembly template.	-	inlbs_mfg_nc.mfg
template_mold_layout	File Storage & Retrieval	Specify the mold layout assembly used as a default template.	-	inlbs_mold_layout.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
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.	-	-
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	Determines whether or not to-tiff exports are compressed using G4 compression	none, g4	none
tiff_type	Data Exchange	Determines the type of tiff items are exported to variables refer to color sets	rgb, palette, grayscale, mono	rgb

Name	Category	Description	Values	Default Value
today's_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	Nominal - Displays dimensions without tolerances. Limits - Displays upper and lower limits. Plusminus - Displays as nominal with plus-minus tolerances. Plusminussym - Displays as nominal with single value for positive and negative tolerance.	nominal, limits, plusminus, plusminussym	limits
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.	-	-
topobus_enable	Data Exchange	Enables usage of the Associative Topology Bus for exchanging data.	yes, no	no
tracesketch	Without Category	Display tracesketches outside of Style feature	no, yes	yes
trail_delay	System	Sets a delay in seconds between trail file steps.	-	0
trail_dir	System	Instructs Pro/ENGINEER to create its trail file in the specified directory rather than in the startup directory.	-	-

Name	Category	Description	Values	Default Value
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
triangulate_filled_areas	Drawing	Subdivide filled areas into triangles (may affect memory usage and plot files).	yes, no	no
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, standard	standard
unattached_geometry_log_path	Without Category	Sets a path to log files with checked objects listing and ghost features details.	-	-
units_system_dic_file	Piping	Specifies the file containing the mapping for MCAT filename and units.	-	\$PTCSRC/text/piping_data/master_catalog/piping_units_system_dic_file.ptd
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

Name	Category	Description	Values	Default Value
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_cadam_plot_data	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_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
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_display_reps	Without Category	Yes to retrieve master rep when graphics rep has no display data.	yes, no	no
use_new_intchg	Assembly	"yes"-Pro/ENGINEER uses the consolidated interchange assembly functionality. "no"-Pro/ENGINEER uses the separate functional interchange assemblies and simplify interchange assemblies.	yes, no	yes

Name	Category	Description	Values	Default Value
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_temp_dir_for_inst	File Storage & Retrieval	Explicitly makes Pro/ENGINEER use the Temp directory for regenerating instances of models.	no, yes	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
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

Name	Category	Description	Values	Default Value
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_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.	-	6
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
vrml_export_version	Data Exchange		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

Name	Category	Description	Values	Default Value
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_simpexp_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
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.	-	\$PTCSRC/html/usage/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
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

Name	Category	Description	Values	Default Value
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 the assembly 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

Name	Category	Description	Values	Default Value
weld_edge_prep_root_o pen	Weld	Specifies an initial default value for the root opening edge preparation.	-	0.25
weld_edge_prep_visibilit y	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_notch_corner_radi us	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_ui_standard	Weld	Specifies standard for welding user interface. ANSI - Use ANSI standards. ISO - Use ISO standards.	iso, ansi	ansi
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
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

Name	Category	Description	Values	Default Value
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

Configuration Option Changes Between Pro/ENGINEER 2001 and Pro/ENGINEER Wildfire 2.0

Name	Status	Category	Description	Values	Default Value
acis_out_version	Changed	Without Category	Allows the user to choose ACIS export version.	public 4, public 5, internal 6, internal 7	5
activate_sketch_with_dflt	New	Features	Activate sketcher automatically using system defaults.	yes, no	no
add_lower_level_components_to_layer	New	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
adobe_distiller	New	Printing & Plotting	Sets the Adobe Distiller command path. Required for PDF and COLOR PDF creation.	-	-
advanced_intersection	New	Assembly	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	no
aec_parameter_file	New	Without Category	Specifies the file name of the user defined piping parameter codes.	-	-
af_copy_references_flag	New	User Interface	Toggle display of Copy References column in Annotation Feature definition dialog	no, yes	no
align_cable_bundles	New	Electromechanical	Align NEWLY CREATED bundles where they meet or branch out	yes, no	yes
allow_move_view_with_move	Changed	Drawing	Disallow the movement of drawing views with the mouse.	yes, no	no
allow_multiple_tool_tips	New	Manufacturing	Allows to use for machining cutting tool with multiple tips.	yes, no	no
allow_ref_scope_change	Changed	Reference control	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_shrink_dimension_before	New	Casting & Mold Design	Allow calculation before relations.	yes, no	no

Name	Status	Category	Description	Values	Default Value
allow_solid_perspective_hlr	New	Model Display	Yes - Allows a solid to be displayed with hidden lines while in perspective.	no, yes	no
allow_workpiece_silhouette	New	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
angle_grid_interval	New	Features	Modify grid space in angular units for handle movement.	-	1
ask_designate_owners	New	Without Category	Show the both required and optional owners to be designated.	yes, no	yes
atb_auto_check_on_activate	New	Data Exchange	Sets state of ATB 'Auto Check Status On Activate' toggle.	on, off	off
atb_auto_check_on_retrieve	New	Data Exchange	Sets state of ATB 'Auto Check Status On Retrieve' toggle.	on, off	on
atb_auto_check_on_update	New	Data Exchange	Sets state of ATB 'Auto Check Status On Update' toggle.	on, off	off
atb_prod_asm_update_by_comp_ids	New	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	New	Without Category	Extends search path for ATB Update and ATB Check Status commands.	-	-
atb_show_log	New	Data Exchange	Sets state of ATB 'Show Log' toggle.	on, off	on
auto_ae_param_file	New	User Interface	Define path and file name of external file used to define parameters that are automatically created in all Annotation Elements.	-	-

Name	Status	Category	Description	Values	Default Value
auto_constr_always_use_offset	Changed	Assembly	Control whether auto constraint should create offsets. Yes - Auto constraint always creates offsets. No - Auto constraint snaps align or mate if surfaces are within tolerance. Never - Auto constraint never create offsets.	yes, no, never	no
auto_regen_views	Changed	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_traceback	New	System	When set to yes, traceback.log file will be generated when fatal error occurs.	yes, no	yes
auto_xml_on_retrieve	Changed	Electromechanical	Automatically load Cabling Logical Reference from an XML file when retrieving the cabling assembly	yes, no	no
auto_xml_on_save	Changed	Electromechanical	Automatically create an XML Logical Reference when saving a cabling assembly	yes, no	no
autobuildz_enabled	New	Drawing	Loads AutobuildZ application.	yes, no	no
autoplace_single_component	New	Assembly	Assembling components with an interface will automatically place the component at the first position satisfying the interface definitions.	no, yes	yes
auxapp_popup_menu_info	New	User Interface	Enable display of popup menu info in the trail file.	yes, no	no
bell	Changed	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

Name	Status	Category	Description	Values	Default Value
blended_transparency	New	Model Display	Yes - Transparent colors will appear using alpha blending (if supported) when the model is shaded.	no, yes	yes
bm_graph_tool	Changed	User Interface	BMX graph tool	public default, public excel_linked, public excel_embedded, internal generic_graphtool	default
bmgr_pref_file	New	User Interface	Specifies the location of the Graph Tool preference file.	-	-
bm_param_restrictions	New	Without Category		yes, no	yes
bump_revnum_on_retrieve_regen	New	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
can_snap_to_missing_ref	Changed	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	Changed	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
centimeter_grid_interval	New	Features	Modify grid space in centimeter units for handle movement.	-	0.1
cgm_use_reversed_image_floats	Changed	Data Exchange	Yes - Enables a metafile to be viewed in Advanced Technology Center's ForReview.	yes, no	yes
check_ghost_features	New	Without Category	Skip features that are already marked non-ghost.	skipchecked, never, always	skipchecked

Name	Status	Category	Description	Values	Default Value
color_windows	Changed	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
comp_angle_offset_eps	New	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
compress_output_files	Changed	File Storage & Retrieval	You can compress object files to store them. Compressed files are slower to read and write, one-half to one-third the size, and fully compatible across systems. Yes - Stores object files in compressed format. No - Object files are not compressed.	yes, no	no
conf_mouse_anim	New	Miscellaneous	Display the presenter's mouse motions when observing a shared Pro/ENGINEER session.	no, yes	yes
copy_dxf_dim_pict	Changed	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
create_temp_interfaces	New	Assembly	Allow the automatic creation of interfaces based on previous assembly instructions.	yes, no	no
def_layer	Changed	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_assem_member, layer_assy_cut_feat, layer_axis, layer_chamfer_feat, layer_comp_design_model, layer_comp_fixture, layer_comp_workpiece, layer_copy_geom_feat, layer_corn_chamf_feat,	-

Name	Status	Category	Description	Values	Default Value
				layer_cosm_round_feat, layer_cosm_sketch, layer_csys, layer_curve, layer_curve_ent, layer_cut_feat, layer_datum, layer_datum_plane, layer_datum_point, layer_detail_item, layer_dgm_conn_comp, layer_dgm_highway, layer_dgm_rail, layer_dgm_wire, layer_dim, layer_draft_constr, layer_draft_dim, layer_draft_dtm, layer_draft_entity, layer_draft_feat, layer_draft_geom, layer_draft_grp, layer_draft_hidden, layer_draft_others, layer_draft_refdim, layer_driven_dim, layer_dwg_table, layer_ext_copy_geom_feat, layer_feature, layer_geom_feat, layer_gtol, layer_hole_feat, layer_intchg_func, layer_intchg_simp, layer_nogeom_feat, layer_nongraph_annot_elem, layer_note, layer_parameter_dim, layer_part_refdim, layer_point, layer_protrusion_feat, layer_quilt, layer_refdim, layer_ribbon_feat, layer_rib_feat, layer_round_feat, layer_sfin, layer_shell_feat, layer_skeleton_model, layer_slot_feat, layer_snap_line, layer_solid_geom, layer_surface, layer_symbol, layer_thread_feat, layer_trim_line_feat, layer_weld_feat	
default_comp_geom_scope	New	Reference control	Set the specified Shared Geometry reference control conditions to newly added components.	all, pg, none	all

Name	Status	Category	Description	Values	Default Value
default_comp_invalid_refs	New	Reference control	Set the specified handling method of out-of-scope Accessible references to newly added components.	copy, prohibit	copy
default_comp_scope_setting	New	Reference control	Set the specified Accessible reference control conditions to newly added components.	all, none, subassemblies, skeleton_model	all
default_comp_skel_geom_scope	New	Reference control	Set the specified Shared Geometry reference control conditions to newly added Skeleton components.	all, pg, none	all
default_comp_skel_scope_setting	New	Reference control	Set the specified Accessible reference control conditions to newly added Skeleton components.	all, none, subassemblies, skeleton_model	all
default_ext_ref_scope	Changed	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_geom_scope	Changed	Reference control	Default value for Geometry Scope allowed for referencing.	all, pg_if_exists, pg_only	all
default_layer_model	New	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_obj_constraints_scope	New	Reference control	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	New	Reference control	Set the specified Shared Geometry reference control conditions to newly created models if template models were not used.	all, pg, none	all

Name	Status	Category	Description	Values	Default Value
default_object_invalid_refs	Changed	Reference control	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	Changed	Reference control	Set default condition for reference control. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all, none, subassemblies, skeleton_model	all
default_placement_scope	Changed	Reference control	Default value for Component Placement Reference filter.	all, intrf_if_exists, intrf_only	all
default_shrink_formula	New	Casting & Mold Design	Default option of shrinkage formula: 1/(1-S) - ASME standard or 1+S (simple).	asme, simple	simple
dim_offscreen_limit	Changed	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
disp_in_open_rep_create	New	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
display_dwg_sketch_constraint	New	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_florin	New	Model Display	Yes: display of the florin (international currency symbol) where possible. No: use the euro symbol exclusively.	no, yes	no

Name	Status	Category	Description	Values	Default Value
display_in_adding_view	Changed	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_tol_by_1000	New	Dimensions & Tolerances	If set to Yes, tolerances for non-angular dimensions will be displayed multiplied by 1000.	yes, no	no
dm_auto_open_zip	New	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_mode	New	Data Management	Indicate which objects will be written to local cache when objects in Pro/E memory are saved.	none, modified, all	all
dm_cache_size	New	Data Management	Enter amount of disk space (in megabytes) to allocate for local file storage.	-	400
dm_http_compression_level	New	Data Management	Set the data compression factor (0-9, 0= no compression) for data exchange with a Windchill server.	-	0
dm_overwrite_contents_on_update	New	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
dm_remember_server	New	Data Management	Primary server/workspace must be set by user for each Pro/E session.	no, yes	yes
dm_secondary_upload	New	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

Name	Status	Category	Description	Values	Default Value
dm_upload_objects	New	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
draw_models_read_only	Changed	Drawing	Yes - Drawing model files are read-only. Driven dimensions, gtols, and similar items cannot be added to the model. Cosmetic_only - Driven dimensions, gtols, cross sections, and explode states can be added to the model, design changes are not allowed.	yes, no, cosmetic_only	no
drawing_view_origin_sys	Changed	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	-
dwg_export_format	Changed	Data Exchange	Provides the ability to choose DWG file version when exporting from Pro/E drawing.	12, 13, 14, 2000	2000
dxblock_to_pro_symbol	New	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
dx_export_format	Changed	Data Exchange	Provides the ability to choose DXF file version when exporting from Pro/E drawing.	12, 13, 14, 2000	2000
dx_in_faceted_brep	New	Data Exchange	Default, faceted boundary representation is allowed in import.	yes, no	yes
dx_out_comments	Changed	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
ecad_export_cuts_as_holes	New	Data Exchange	Exports Pro/ENGINEER cuts as holes to ECAD systems.	yes, no	yes

Name	Status	Category	Description	Values	Default Value
ecad_missing_component_status	New	Data Exchange	Setup default status for missing components in "Investigate Placement" Dialog.	keep_missing, delete_missing	keep_missing
ecad_other_outlines_def_name	Changed	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_pin_hole_import	New	Data Exchange	Setup the default for the import of ECAD Holes from type PIN (IDF 3.0 ONLY)	yes, no	no
ecad_via_hole_import	New	Data Exchange	Setup the default for the import of ECAD Holes from type VIA (IDF 3.0 ONLY)	yes, no	yes
enable_face_preselection	New	Without Category		yes, no	yes
enable_implied_joints	New	Assembly	Allow underconstrained components as mechanism connections.	yes, no	yes
enable_k01_measure_analysis	New	Without Category	Enable or disable new UI for Measure Analysis.	yes, no	no
enable_offset_fan_curve	New	Without Category	Enables fan curve option for offset tool.	yes, no	no
enable_protk_xml_schema	New	Application Programming Interfaces	Enables XML-Schema validation for Pro/TK feature XMLs	yes, no	no
enable_tree_independent	New	System	Setup whether tree customization will allow separating the tree window from Pro/E window.	yes, no	no
expand_all_with_nested_layers	New	Without Category	Yes - expand all including nested sub layers.	yes, no	yes
fan_significant_locations_only	New	Electromechanical	Setup whether only significant locations should be flattened on flattening a harness.	no, yes	yes

Name	Status	Category	Description	Values	Default Value
fan_with_network	New	Electromechanical	Setup whether the network should be included on flattening a harness.	yes, no	yes
fast_highlight	New	Model Display	Yes - Use simplified highlighting which redisplayes more quickly at the expense of certain details. This option may increase memory usage slightly.	no, yes	no
fem_allow_named_mesh_files	New	FEM	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_default_solver	Changed	FEM	Specifies default solver.	ansys, nastran	-
fem_mesh_preserve	Changed	FEM	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_solid_shell_auto_constraint	Changed	FEM	Yes - SPCs for the rotational DOF will automatically be created at the interfacing nodes between solids and shells. That is shell rotations will be fixed. No - No such constraints will automatically be created. Ask - Prompt the user before running.	yes, no, ask	no
file_open_default_folder	Changed	File Storage & Retrieval	Sets default dir for File>Open. Working_directory - Search working directory. In_session - Search objects in session. Pro_library - Use only if Pro/LIBRARY is installed. Workspace - Use only with Pro/INTRALINK. Commonsplace - Use only with Pro/INTRALINK.	default, working_directory, in_session, pro_library, my_documents	-

Name	Status	Category	Description	Values	Default Value
folder_display_file_limit	New	File Storage & Retrieval	Sets the default number of files that will be displayed when viewing the contents of a folder using the Folder Navigator.	-	-1
foot_grid_interval	New	Features	Modify grid space in foot units for handle movement.	-	0.083333
force_upd_assem_m p_in_simp_rep	New	Assembly	Update mass properties parameters even when some of the components of the assembly are not in master rep.	yes, no	no
general_undo_stack_limit	New	System	Setup the default number of entries in the top level Undo/Redo (default = 50).	-	50
highlight_layer_items	New	Layers	Specifies whether items selected in the layer tree should be highlighted in the graphics area.	yes, no	yes
hole_diameter_override	Changed	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_parameter_file_path	Changed	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.	-	-
iges_out_assembly_default_mode	Changed	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 geom info of comps and assy feats.	public flat, internal one level, internal all levels, public one_level, public all_levels, internal all parts, public all_parts	flat

Name	Status	Category	Description	Values	Default Value
ignore_all_ref_scope_settings	Changed	Reference control	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
ignore_export_geometry_declaration	Changed	Reference control	Ignore Export Geometry settings.	yes, no	no
inch_grid_interval	New	Features	Modify grid space in inch units for handle movement.	-	0.125
info_output_format	New	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.	public text, public html, internal xml_text, internal dbg_text	html
instance_search_exhaustive	Changed	File Storage & Retrieval	Specifies extent of instance search. Yes - Include file type when entering instance name in File>Open: type in input panel (e.g. inst_name.prt) or select in drop-down list. If no .idx files are found, only working dir is searched; each generic is not.	no, yes	no
intf_cadds_version	New	Data Exchange	Allows the user to switch between the default and alternate converter version for interface with CADDSS5	internal 10, internal 11, public 12, public 13	-
intf_in_dwg_view	Changed	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_granite_direct_enable	New	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

Name	Status	Category	Description	Values	Default Value
intf_in_use_template_models	New	Data Exchange	Allows the user to use template models in Pro/INTERFACE import.	yes, no	no
intf_out_cat_start_model	New	Data Exchange	Specifies Catia start model to be used for CATIA II .model export.	-	-
intf_out_max_bspline_degree	Changed	Data Exchange	Controls the maximum degree of exported B-spline surfaces when exporting through IGES	-	16
intf_use_variable_size	New	Data Exchange	Yes - IGES,DXF,STEP files with variable sized sheet information are imported and put on the appropriate variable-size format. Without a sheet size, the system tries to apply the correct variable-size format. No - The IGES,DXF,STEP drawing is placed on a standard sized format.	no, yes	yes
intf2d_iges_out_hatch	New	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	New	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	New	Data Exchange	Create a single multi-line note on importing multi-line text when is set to YES.	yes, no	yes
intf2d_out_acad_mtext	New	Data Exchange	Create a single MTEXT entity for multi-line text notes when is set to YES.	yes, no	yes
intf2d_out_acad_text_align	New	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

Name	Status	Category	Description	Values	Default Value
intf2d_out_cgm_ver	New	Data Exchange	Provides the ability to choose CGM metafile version when exporting from Pro/E drawing.	1, 3	1
intf2d_out_open_log_window	New	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	New	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
intf3d_ideas_import_filter	New	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	New	Data Exchange	Specifies the path for I-DEAS installation. Use the full pathname to avoid problems.	-	-
intf3d_ideas_run_command	New	Data Exchange	Specifies the I-DEAS run command. Default command is "ideas".	-	ideas.cmd
intf3d_out_cat2_ident_crv	Changed	Data Exchange	Specifies model parameter IDENT_CRV while exporting to Catia.	-	-1
intf3d_out_cat2_ident_pt	Changed	Data Exchange	Specifies model parameter IDENT_PT while exporting to Catia.	-	-1
intf3d_out_cat2_infinity	Changed	Data Exchange	Specifies model parameter INFINITY while exporting to Catia.	-	-1
intf3d_out_cat2_model_sz	Changed	Data Exchange	Specifies model parameter MODEL_SZ while exporting to Catia.	-	-1
intf3d_out_cat2_sag	Changed	Data Exchange	Specifies model parameter SAG while exporting to Catia.	-	-1
intf3d_out_cat2_step	Changed	Data Exchange	Specifies model parameter STEP while exporting to Catia.	-	-1
intf3d_parasolid_export_schema	New	Without Category		-	sch_10004
intf3d_ug_install_dir	New	Without Category	Specifies the path for Unigraphics installation. Use the full pathname to avoid problems.	-	-

Name	Status	Category	Description	Values	Default Value
isogen_attribute_map_file	New	Piping	Specifies the mapping between isogen parameter and attribute name.	-	\$PTCSRC/text/piping_data/isodata/isogen_attribute_map.ptd
jlink_java_command	New	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	New	Application Programming Interfaces	Option to force use of Java-2 JRE command line ("java" instead of "jre")	off, on	on
log_dwg_spr1150166_detection	New	Without Category	Create a log file and include if a particular drawing needs correction for SPR #1150166.	no, yes	no
mapkey	Changed	Miscellaneous	Stores a keyboard macro. Use Tools>Mapkeys to create, edit and delete mapkeys.	-	-
mass_property_calculate	New	Environment	Calculate mass properties upon regeneration or by user request.	by_request, automatic	by_request
mdo_cl_executable	New	Mechanism	Path for Custom Load executable.	-	-
mdo_density	Changed	Mechanism	Mass density for dynamics.	-	0
mdo_enable_joint_friction	New	Mechanism	Enable joint axis friction.	yes, no	no
mdo_explicit_integration	New	Mechanism	Enable explicit integration for dynamic analyses.	yes, no	no
mdo_spring_wire_radius	Changed	Mechanism	Spring wire radius for dynamics spring icons.	-	0.15
mdo_use_mm_body_order	New	Mechanism	Use Mechanica Motion body order. (Make the ground to be the first body in connection).	yes, no	no
meter_grid_interval	New	Features	Modify grid space in meter units for handle movement.	-	0.001

Name	Status	Category	Description	Values	Default Value
mfg_auto_retract_clear_dist	New	Without Category		-	-0.001
mfg_process_print_dir	New	Without Category	Specifies the directory where Mfg Process html page get stored. Use the full pathname to avoid problems.	-	-
mfg_process_table_setup_dir	New	Without Category	Specifies the directory where Mfg Process Table settings get stored. Use the full pathname to avoid problems.	-	-
mfg_process_template_dir	New	Without Category	Specifies the directory where Mfg Process Templates get stored. Use the full pathname to avoid problems.	-	-
mfg_start_model_dir	New	File Storage & Retrieval	Provides the complete path to the directory containing mfg start assemblies.	-	-
mfg_workpiece_transparency	New	Manufacturing		-	0.5
millimeter_grid_interval	New	Features	Modify grid space in millimeter units for handle movement.	-	1
minimum_angle_dimension	New	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_reference_scope_change	Changed	Reference control	Yes - Users can change the scope setting of components.	yes, no	yes
mp_calc_level	New	Without Category	Specifies the level of mass properties calculation.	assembly_only, all_models	assembly_only
nc_jobman_visible	New	Without Category	Yes - make "Submit Path" and "Abort Path" visible. No - make "Submit Path" and "Abort Path" invisible.	yes, no	no

Name	Status	Category	Description	Values	Default Value
new_asm_regen_rev_nums	Changed	Assembly	Yes - Regenerate command will not increase the model revision number of the top assembly unless features belonging to the top assembly actually get regenerated. Regenerate command will increase model revision numbers of subassemblies if features in the subassembly regenerate. No - Regenerate command will always increase the revision number of the top assembly and almost never increase the revision numbers of subassemblies, regardless of whether or not assembly features regenerate.	yes, no	yes
new_designate_ui	New	Without Category		yes, no	yes
new_parameter_ui	Changed	User Interface	Enable the new parameter editor and user interface.	yes, no	yes
new_relation_ui	Changed	User Interface	Enable the new relations editor and user interface.	yes, no	yes
number_user_colors	Changed	Colors	Specifies the maximum number of different wireframe colors that can display in the graphics area(s) at any given moment.		-1
orientation	Changed	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	New	Without Category	Set spin orientation style to dynamic or anchored.	anchored, dynamic	dynamic
output_mfg_xml	New	Without Category	Allows export/import mfg model to/from xml file. Yes - make Export/Import options visible. No - make them invisible.	yes, no	no

Name	Status	Category	Description	Values	Default Value
overlays_enabled	Changed	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	yes
param_dec_places	New	User Interface	Sets the number of decimal places (0-14) to display in parameters.	-	6
pdf_plot_config	New	Printing & Plotting	Set the pcf filename to be used when saving to PDF format.	-	-
pen1_line_weight	Changed	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
pen2_line_weight	Changed	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	Changed	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	Changed	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	Changed	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	Changed	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	Changed	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

Name	Status	Category	Description	Values	Default Value
pen8_line_weight	Changed	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	New	Photorender	Allows the user to disable warnings about PhotoLux only capabilities.	yes, no	yes
photorender_default_height	Changed	Photorender	Sets the default height (in pixels) for the custom size rendered output	-	450
photorender_default_width	Changed	Photorender	Sets the default width (in pixels) for the custom size rendered output	-	600
photorender_memory_usage	Changed	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
pipe_bend_locations_csys_units	Changed	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_solid_label_format	New	Piping	Specify the pipe solid label format that will be used to automatically generate pipeline labels.	-	size,- ,specification,- ,mnemonic,- ,number,- ,insulation
pipeline_assembly_library_dir	New	Piping	Sets the default pipeline assembly library directory.	-	\$PTCSRC/text/ piping_data/pipeline lib
pipeline_solid_start_part_name	New	Piping	Sets the default pipeline solid start part name.	-	pipeline_solid_start.prt
pipeline_spool_label_format	Changed	Piping	Specify the pipeline spool label format that will be used to generate the spool labels during pipeline spool generation.	-	mnemonic- number- prefix[0]spool num[01]suffix[1]

Name	Status	Category	Description	Values	Default Value
pipng_bolt_nut_select_file	New	Piping	Piping bolts and nuts selection file.	-	\$PTCSRC/text/piping_data/master_catalog/piping_bolt_nut_select.ptd
pipng_extend_dim_scheme	New	Piping	Specify a dimension scheme for piping extend segment.	cartesian, cylindrical, spherical	spherical
pipng_joint_fitting_clearance	New	Piping	Sets the piping joint minimum fitting clearance value.	-	1
pipng_joint_fitting_offset	New	Piping	Sets the piping joint minimum fitting offset value.	-	1
pipng_min_bolting_clearance	Changed	Piping	Sets the default piping minimum bolting clearance value.	-	1
pipng_schematic_driven	New	Piping	Set the Schematic Driven Pipeline Modeling ON/OFF	no, yes	no
pipng_schematic_xml_dir	New	Piping	Specify directory with full path containing schematic information XML file(s).	-	c:/tmp/dump
pipng_wildfire_convert	New	Piping	Always ask before converting old piping assemblies to update the network structure.	manual, never, automatic	manual
prehighlight	Changed	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.	internal none, internal all, internal geom_only, public yes, public no	-
pro_cbltrm_dir	New	Electromechanical	Sets the directory from which terminators are retrieved. Use the full pathname to avoid problems.	-	-

Name	Status	Category	Description	Values	Default Value
pro_editor_command	Changed	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 drawing_file_editor and/or part_table_editor and/or relation_file_editor.	-	-
pro_palette_dir	New	Drawing	Sets the default directory for drawing symbol palette files.	-	\$PTCSRC/symbols/palette
pro_stheno_command	New	Data Exchange	Specifies the command for Stheno launch.	-	-
pro_unit_sys	Changed	Environment	Specifies the default system of units for new models.	mks, cgs, mmns, fps, ips, proe_def, mmks	-
provide_pick_message_always	Changed	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
ratio_grid_interval	New	Features	Modify grid space in percentage units for handle movement.	-	0.05
read_parameters_of_excluded	New	Data Management	Allow Pro/ENGINEER designated parameters on excluded objects in Simplified Representation to be visible in Model Tree columns.	yes, no	no
ref_scope_copy_color	Changed	Reference control	Set color of backedup references.	-	55
ref_scope_copy_color_change	Changed	Reference control	Enable changing colors of backedup references during selection.	no, yes	yes
ref_scope_no_pick_to_copy	Changed	Reference control	Disable selecting references backedup by references control.	no, yes	no
ref_scope_no_pick_to_prohibit	Changed	Reference control	Disable selecting references prohibited by references control.	no, yes	yes
ref_scope_prohibit_color	Changed	Reference control	Set color of prohibited references.	-	45

Name	Status	Category	Description	Values	Default Value
ref_scope_prohibit_color_change	Changed	Reference control	Enable changing colors of prohibited references during selection.	no, yes	yes
relation_text_trail_output	New	Miscellaneous	No - Disable trail output from Text layout in Relation Parameter UI if it hasn't been changed.	yes, no	yes
relations_num_constants_units	New	Without Category	Specifies whether diagnostic should be issued when an unitless numeric value causes relation expression evaluation trouble.	yes, no	no
relations_units_sensitive	New	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	New	Without Category	Yes - Sets the default of the ATTACH TYPE menu of the last options picked.	yes, no	yes
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.	-	1
restricted_values_definition	New	Without Category	Specify name and path of the file contained restricted values definition.	-	-
retrieve_data_sharing_ref_parts	Changed	File Storage & Retrieval	Automatic reference parts retrieval for dependent data sharing features.	yes, no, ignore_missing	no
retrieve_merge_ref_parts	Changed	Without Category	Automatic reference parts retrieval for dependent Merge.	yes, no, ignore_missing	no
reverse_spiral_finishing_type	New	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.	-	0

Name	Status	Category	Description	Values	Default Value
save_hidden_items_w_status	New	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	Changed	File Storage & Retrieval	With family tables of solid parts, determines how instances are saved. None - Instance accelerator files not used. Explicit - Save instance accelerator files when instances explicitly saved. Always - Always save instance accelerator files.	none, explicit, always	none
save_model_display	Changed	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 View Performance dialog (shaded).	public wireframe, public shading_low, public shading_high, public shading_lod, internal no_display	shading_lod
scope_invalid_refs	Changed	Reference control	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
section_color	Changed	Sketcher	Specifies the color of sketched sections.	default, drawing_color	-
select_on_dtm_edges	Changed	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
show_all_mold_layout_buttons	New	Casting & Mold Design	Sets all mold layout buttons visible.	yes, no	no
sim_additive_mass	New	Simulation	Treats Mass Idealizations as additive if they are applied to the same entity and suppresses Mass precedence rules.	yes, no	no

Name	Status	Category	Description	Values	Default Value
sim_asm_modeling	New	Simulation	Controls automatic creation of assembly links and free connections in Mechanica. The default is YES.	yes, no	yes
sim_beamsection_path	New	FEM	Specifies the path where the Beam Sections will be stored.	-	-
sim_display_agem_controls	New	Simulation	Toggles display of AutoGEM Controls.	yes, no	yes
sim_display_arrow_scale	Changed	Simulation	Toggles automatic arrow scaling.	yes, no	yes
sim_display_arrow_tail_touching	New	Simulation	Toggles display of Load arrows to have tails or heads touching.	yes, no	no
sim_display_beam_releases	New	Simulation	Toggles display of Beam Release icons.	yes, no	yes
sim_display_beam_sections	Changed	Simulation	Toggles display of beam section icons.	yes, no	yes
sim_display_beams	Changed	Simulation	Toggles display of beam icons.	no, yes	yes
sim_display_contact_regions	New	Simulation	Toggles display of Contact Region icons.	yes, no	-
sim_display_fasteners	New	Simulation	Toggles display of Fastener icons.	yes, no	-
sim_display_in_spin	New	Simulation	Toggles display of simulation objects in Dynamic Spin.	yes, no	yes
sim_display_interfaces	New	Without Category	Toggles display of structure connection icons.	no, yes	yes
sim_display_load_colors	New	Simulation	Toggles display of Load colors.	yes, no	yes
sim_display_load_distribution	Changed	Simulation	Toggles display of distributed load vectors over entire entity.	yes, no	yes
sim_display_load_icons	Changed	Simulation	Toggles display of load icons.	yes, no	yes
sim_display_load_value	New	Simulation	Toggles display of Load values.	yes, no	yes
sim_display_material_assignments	New	Simulation	Toggles display of Material Assignment icons.	yes, no	yes

Name	Status	Category	Description	Values	Default Value
sim_display_measures	New	Simulation	Toggles display of Simulation Measure icons.	yes, no	yes
sim_display_mesh_and_model	New	Simulation	Toggles display of Mesh together with Model.	yes, no	no
sim_display_mesh_controls	New	Simulation	Toggles display of Mesh Controls.	yes, no	yes
sim_display_mesh_entities	New	Simulation	Toggles display of Mesh entities.	yes, no	yes
sim_display_mesh_mode	New	Simulation	Specifies the mode of mesh display in FEM preprocessor. Values: NO_MESH, WIREFRAME, HIDDEN, NOHIDDEN, SHADING.	no_mesh, wireframe, hidden, nohidden, shading	wireframe
sim_display_mesh_quality	New	Simulation	Specifies the quality of mesh display in Mechanica. Values: FINE, MEDIUM, COARSE.	fine, medium, coarse	coarse
sim_display_mesh_shells_thick	New	Simulation	Toggles the display of shell thickness in Mechanica mesh. YES - Display shells with real thickness. NO - Display shells with zero thickness.	no, yes	no
sim_display_mesh_shrink_elems	New	Simulation	Controls display of Mesh through a specified percentage of shrinking of its elements.	-	-1
sim_display_motion_cams	Changed	Simulation	Toggles display of motion cam icons.	yes, no	yes
sim_display_motion_connections	Changed	Simulation	Toggles display of motion connection icons.	yes, no	yes
sim_display_motion_contact_rgns	Changed	Simulation	Toggles display of motion contact region icons.	yes, no	yes
sim_display_motion_design_vars	Changed	Simulation	Toggles display of design variable icons.	yes, no	yes
sim_display_motion_drivers	Changed	Simulation	Toggles display of motion driver icons.	yes, no	yes

Name	Status	Category	Description	Values	Default Value
sim_display_motion_gears	Changed	Simulation	Toggles display of motion gear icons.	yes, no	yes
sim_display_motion_ground_pnts	Changed	Simulation	Toggles display of motion ground point icons.	yes, no	yes
sim_display_motion_lcs	Changed	Simulation	Toggles display of local coordinate systems icons.	yes, no	yes
sim_display_motion_loads	Changed	Simulation	Toggles display of motion load icons.	yes, no	yes
sim_display_motion_slots	Changed	Simulation	Toggles display of motion slot icons.	yes, no	yes
sim_display_names	New	Simulation	Toggles display of Simulation Object names.	yes, no	no
sim_display_rigid_dof	New	Simulation	Toggles display of Rigid Link DOF icons.	yes, no	-
sim_display_rigid_links	New	Simulation	Toggles display of Rigid Link icons.	yes, no	-
sim_display_shells	Changed	Simulation	Toggles display of shell icons.	no, yes	yes
sim_display_spot_welds	Changed	Simulation	Toggles display of spot weld icons.	yes, no	yes
sim_display_therm_bcs	New	Simulation	Toggles display of Thermal Boundary Condition icons.	yes, no	-
sim_display_weighted_dof	New	Simulation	Toggles display of Weighted Link DOF icons.	yes, no	-
sim_display_weighted_links	New	Simulation	Toggles display of Weighted Link icons.	yes, no	-
sim_display_welds	New	Simulation	Toggles display of Weld icons.	yes, no	yes
sim_fatigue_biaxiality_correct	New	Simulation	Controls the use of biaxiality correction. The default value is YES.	yes, no	yes
sim_fatigue_biaxiality_method	New	Simulation	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

Name	Status	Category	Description	Values	Default Value
sim_fatigue_confidence_level	New	Simulation	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	New	Simulation	Controls the use of external material data. The default value is NO.	no, yes	no
sim_fatigue_hysteresis_gate	New	Simulation	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	New	Simulation	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	New	Simulation	Controls the application of the mean stress correction. The default value is YES.	yes, no	yes
sim_fatigue_mean_stress_method	New	Simulation	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
sim_fatigue_safety_margin	New	Simulation	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	New	Simulation	Specifies the directory for user files (for example, external material files). The default value is the current directory.	-	-
sim_fatigue_write_surface_stress	New	Simulation	Controls writing of surface stresses to a neutral file. The default value is NO.	no, yes	no
sim_fem_nastran_use_pshell	New	Simulation	Yes - use PSHELL cards, No - use PCOMP cards for laminate composite shells in NASTRAN output file.	yes, no	no

Name	Status	Category	Description	Values	Default Value
sim_html_report_prefs_file	New	FEM	Specifies the path to the Simulation post processor HTML report preferences file.	-	-
sim_load_mech_mesh	New	Simulation	No - The user is prompted to load the existing Mechanics mesh file from the current working directory. Yes - The mesh file, if exists, is loaded automatically and the mesh is displayed.	no, yes	yes
sim_mat_poissons_notation	New	Simulation	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_memory_usage	New	Simulation	Specifies the amount of memory in megabytes to be used in Mechanics run and FEM mesher. The default value is 128.	-	128
sim_motion_analysis_accuracy	New	Simulation	Specifies the default accuracy to which Mechanics determines values at each increment of the analysis. This number represents the acceptable fractional error in the computation of joint axis positions and rates at each time increment of the analysis. The default value is 0.0001.	-	0.0001
sim_motion_analysis_assem_tol	New	Simulation	Specifies the default error allowed for the assembly analysis. The default value is 0.001.	-	0.001
sim_motion_analysis_duration	New	Simulation	Specifies the default length of time in seconds during which the analysis runs. The analysis duration can be less than the duration of the motion simulation. The default value is 10.0.	-	10

Name	Status	Category	Description	Values	Default Value
sim_motion_analysis_increment	New	Simulation	Specifies the default increment between steps at which Mechanica reports the motion of the mechanism. The default value is 0.1	-	0.1
sim_motion_analysis_integrator	New	Simulation	Controls the default integrator Mechanica uses when running the motion analysis. The value types are IMPLICIT (default) and EXPLICIT.	implicit, explicit	implicit
sim_motion_analysis_method	New	Simulation	Controls the default method Mechanica uses to analyze the motion of your model. The default is FASTEST_AVAILABLE.	fastest_available, time_integration, kinetostatic	fastest_available
sim_motion_analysis_start_time	New	Simulation	Specifies the default time in seconds at which the motion analysis begins. The default value is 0.	-	0
sim_motion_analysis_vel_tol	New	Simulation	Specifies the default error allowed for the velocity analysis. The default value is 0.001.	-	0.001
sim_motion_output_fly_file	New	Simulation	Controls the creation of .fra file when saving results for a motion model, The .fra file is used to import a motion animation into Pro/FLY-THROUGH. The default is NO.	no, yes	no
sim_nastran_use_coupmass	New	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	New	Simulation	Outputs FEM mesh element IDs on each layer into XML file.	yes, no	no
sim_output_obj_names	New	Without Category	Output Beam Sections names, Analyses names, Coordinate Systems names as comments.	yes, no	no

Name	Status	Category	Description	Values	Default Value
sim_pp_background_color	New	Simulation	Specifies the background color of the work area for the results display. The default value is PROE.	proe, blue, black, white	proe
sim_pp_path_absolute	New	Simulation	Choose Absolute (Yes) or Relative (No) path to "rwd" file.	yes, no	yes
sim_pp_template_dir	New	Simulation	Sets the default directory for results template files.	-	-
sim_pp_vector_plot_arrow_2d	New	Simulation	"Yes" plots 2D arrows, "NO" plots 3D arrows in Vector Plot of Mechanics Postprocessor.	yes, no	no
sim_pp_vrml_export_format	New	Simulation	Flags to denote VRML1.0 or VRML2.0 (default) output.	vrml2.0, vrml1.0	vrml2.0
sim_pp_vrml_feature_edges	New	Simulation	Flag to enable the inclusion of feature edges in the VRML file. The default value is NO.	no, yes	no
sim_run_copy_fem_neutral_file	New	Simulation	Controls whether the FEM neutral file (with extension .fnf) is copied into the study directory. The default value is YES.	yes, no	yes
sim_run_out_dir	New	Simulation	Specifies the directory for Mechanics run output storage. By default, the current directory is used.	-	-
sim_run_tmp_dir	New	Simulation	Specifies the directory for Mechanics run temporary storage. By default, the current directory is used.	-	-
sim_smooth_aspect_ratio	New	Simulation	Sets the element aspect ratio criterion for the mesh smoothing process.	-	7
sim_smooth_edge_angle	New	Simulation	Sets the element edge angle criterion (Quadrilateral elements only), measured in degrees, for the mesh smoothing process.	-	30

Name	Status	Category	Description	Values	Default Value
sim_smooth_skew	New	Simulation	Sets the element skew angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	45
sim_smooth_taper	New	Simulation	Sets the element taper criterion (Quad elements only) for the mesh smoothing process.	-	0.5
sim_smooth_warp_angle	New	Simulation	Sets the element warp angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	10
sim_use_layers	New	Simulation	Allows layers to become groups for Mechanics Independent mode.	no, yes	no
simulation_fem_mode	Changed	Simulation	Yes - Activates FEM mode. No - Allows access to Mechanics only. Prompt (default) - Brings Mechanics 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	New	Simulation	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	New	Simulation	If set to yes, the Process Guide will be launched automatically upon entry into Mechanics, if template files are present.	yes, no	no
simulation_process_template_dir	New	Simulation	Specifies the default location of the Process Guide template files.	-	-
simulation_product	New	Simulation	Set Mechanics product being selected (Structure/Thermal/Motion). Prompt (default) - Brings Mechanics Model Type Dialog. This option is used only if information about Mechanics product is not yet stored with model.	public structure, public thermal, internal mold, internal motion, public prompt	prompt

Name	Status	Category	Description	Values	Default Value
sketcher_collinear_skamps	New	Sketcher	Yes - collinear constraints will be used by Intent Manager; No - collinear constraints will not be used by Intent Manager.	no, yes	yes
sketcher_disp_weak_dimensions	New	Sketcher	Yes - weak dimensions will be displayed; No - weak dimensions will not be displayed.	yes, no	yes
sketcher_equal_length_skamps	New	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_skamps	New	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	New	Sketcher	Enter a grid angle value to override the default grid angle value.	-	0
sketcher_import_exact_geom	Changed	Sketcher	During import in sketcher use exact geometry.	yes, no	yes
sketcher_lineup_horizontal_skamps	New	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_vertical_skamps	New	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	New	Sketcher	Yes - modified dimensions will be locked; No - modified dimensions will not be locked.	yes, no	no
sketcher_midpoint_skamps	New	Sketcher	Yes - midpoint constraints will be used by Intent Manager; No - midpoint constraints will not be used by Intent Manager.)	no, yes	yes
sketcher_num_digits	New	Sketcher	Enter a number of digits for scetcher accuracy.	-	2

Name	Status	Category	Description	Values	Default Value
sketcher_parallel_skamps	New	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_skamps	New	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_entry_skamps	New	Sketcher	Yes - point on entry constraints will be used by Intent Manager; No - point on entry constraints will not be used by Intent Manager.	no, yes	yes
sketcher_rel_accuracy	New	Sketcher	Enter a sketcher relative accuracy.	-	-1
sketcher_same_point_skamps	New	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	New	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	New	Sketcher	"Automatic" - grid spacing will be automatically set, "Manual" - grid spacing will be manually set.	automatic, manual	automatic
sketcher_set_grid_x_spacing	New	Sketcher	Enter a x-grid spacing value to override the default x-grid spacing value.	-	1
sketcher_set_grid_y_spacing	New	Sketcher	Enter a y-grid spacing value to override the default y-grid spacing value.	-	1
sketcher_symmetric_skamps	New	Sketcher	Yes - symmetric constraints will be used by Intent Manager; No - symmetric constraints will not be used by Intent Manager.	no, yes	yes

Name	Status	Category	Description	Values	Default Value
sketcher_tangent_skamps	New	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_stack_limit	Changed	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	Changed	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
smt_bend_notes_default_display	New	Sheetmetal	Define the default state of Bend Note display (sheetmetal Mode)	yes, no	yes
smt_bend_notes_direction_down	New	Sheetmetal	Define the Down Direction symbol for Bend Notes (sheetmetal Mode)	-	default
smt_bend_notes_direction_up	New	Sheetmetal	Define the Up Direction symbol for Bend Notes (sheetmetal Mode)	-	default
smt_bend_notes_order	New	Sheetmetal	Define the Fields Order for Bend Notes (sheetmetal Mode)	-	&type&direction&angle
smt_bend_notes_type_formed	New	Sheetmetal	Define the Formed symbol for Bend Notes (sheetmetal Mode)	-	default
smt_bend_notes_type_rolled	New	Sheetmetal	Define the Rolled symbol for Bend Notes (sheetmetal Mode)	-	default
smt_mp_method	Changed	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

Name	Status	Category	Description	Values	Default Value
spherical_map_size	New	Drawing	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_rate_zoom_sensitive	Changed	Model Display	Allows slower spinning when zoomed in on a model.	yes, no	no
spin_with_orientation_center	New	Without Category	Display orientation center while reorienting.	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.	public ap203_is, internal ap203_dis, public ap214_cd, public ap202_is, public 203_is_ext, public ap214_dis, public ap209_dis, public ap214_is	ap203_is
style_default_connections	New	Without Category	Yes - Add connections to a newly created Style surface.	yes, no	yes
system_hidden_color	Changed	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
tbl_driven_tol_value_edit	New	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	Changed	File Storage & Retrieval	Specifies the designated template assembly. Use full path to avoid problems.	-	inlbs_asm_design.asm

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.mfg
template_mfgcmm	New	File Storage & Retrieval	Specify the model used as the default manufacturing cmm template.	-	inlbs_mfg_cmm.mfg
template_mfgemo	New	File Storage & Retrieval	Specify the model used as the default manufacturing expert machinist template.	-	inlbs_mfg_emo.mfg
template_mfgmold	Changed	File Storage & Retrieval	Specify the model used as the default manufacturing mold template.	-	inlbs_mfg_mold.mfg
template_mfgnc	New	File Storage & Retrieval	Specify the model used as the default manufacturing assembly template.	-	inlbs_mfg_nc.mfg
template_mold_layout	Changed	File Storage & Retrieval	Specify the mold layout assembly used as a default template.	-	inlbs_mold_layout.asm
tracesketch	New	Without Category	Display tracesketches outside of Style feature	no, yes	yes
triangulate_filled_areas	New	Drawing	Subdivide filled areas into triangles (may affect memory usage and plot files).	yes, no	no
ui_theme	New	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, standard	standard
unattached_geometry_log_path	New	Without Category	Sets a path to log files with checked objects listing and ghost features details.	-	-
units_system_dictionary	New	Piping	Specifies the file containing the mapping for MCAT filename and units.	-	\$PTCSRC/text/piping_data/master_catalog/piping_units_system_dictionary.ptd
update_rep_refs	Changed	Assembly	"Yes" updates top assembly simplified reps for replacements during regeneration and upon replacement.	yes, no	yes

Name	Status	Category	Description	Values	Default Value
use_export_2d_dialog	New	Data Exchange	Yes - Opens export options dialog when exporting Pro/E drawing. No - exports file without the options dialog.	yes, no	yes
use_pre_wildfire_text_font	New	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
user_defined_grid_interval	New	Features	Modify grid space in user defined length units for handle movement.	-	0.5
verify_on_save_by_default	New	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
visible_message_lines	Changed	User Interface	Sets the default number of visible message lines in the Pro/ENGINEER message area.	-	6
vrmlexport_version	New	Data Exchange		2.0, 1.0	2
web_browser_history_days	New	System	Enter number of days to store history records.	-	20
web_browser_homepage	New	System	Enter location of Pro/E browser home page.	-	\$PTCSRC/html/usascii/pro/helpsystop/resource_center_index.htm
web_disable_js_command	New	Application Programming Interfaces	Enter a specific javascript command to be disabled	-	-
web_enable_javascript	New	Without Category	Enables or disables Pro/Web.Link in the embedded browser.	on, off	off

Name	Status	Category	Description	Values	Default Value
web_enable_subscription	New	Application Programming Interfaces	Enable the subscription mechanism for Connections	no, yes	yes
web_max_mail_size	New	Without Category	Set the maximum email attachment size in kilobytes.	-	5000
weld_color	Changed	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_notch_corner_radius	New	Weld	Specifies an initial default value for the radius at weld notch corner.	-	0.1
weld_notch_height	New	Weld	Specifies an initial default value for the weld notch height.	-	0.4
weld_notch_radius	New	Weld	Specifies an initial default value for the weld notch radius.	-	0.5
weld_notch_width	New	Weld	Specifies an initial default value for the weld notch width.	-	0.5
windows_scale	Changed	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

Configuration Option Changes Between Pro/ENGINEER Wildfire and Pro/ENGINEER Wildfire 2.0

Name	Status	Category	Description	Values	Default Value
add_lower_level_comps_to_layer	New	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
adobe_distiller	New	Printing & Plotting	Sets the Adobe Distiller command path. Required for PDF and COLOR PDF creation.	-	-
advanced_intersection	New	Assembly	Control the appearance of the intersection tool and scope of available functionalities.	yes, no	no
af_copy_references_flag	New	User Interface	Toggle display of Copy References column in Annotation Feature definition dialog	no, yes	no
allow_multiple_tool_tips	Changed	Manufacturing	Allows to use for machining cutting tool with multiple tips.	yes, no	no
allow_shrink_dim_before	New	Casting & Mold Design	Allow calculation before relations.	yes, no	no
ask_designate_owners	New	Without Category	Show the both required and optional owners to be designated.	yes, no	yes
atb_prod_asm_upd_by_component_ids	New	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	Changed	Without Category	Extends search path for ATB Update and ATB Check Status commands.	-	-
auto_ae_param_file	New	User Interface	Define path and file name of external file used to define parameters that are automatically created in all Annotation Elements.	-	-

Name	Status	Category	Description	Values	Default Value
auto_constr_always_use_offset	Changed	Assembly	Control whether auto constraint should create offsets. Yes - Auto constraint always creates offsets. No - Auto constraint snaps align or mate if surfaces are within tolerance. Never - Auto constraint never create offsets.	yes, no, never	no
auto_regen_views	Changed	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_traceback	New	System	When set to yes, traceback.log file will be generated when fatal error occurs.	yes, no	yes
autobuildz_enabled	New	Drawing	Loads AutobuildZ application.	yes, no	no
auxapp_popup_menu_info	New	User Interface	Enable display of popup menu info in the trail file.	yes, no	no
bmx_param_restrictions	New	Without Category		yes, no	yes
bump_revnum_on_retrieve_regen	Changed	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
color_windows	Changed	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
conf_mouse_anim	Changed	Miscellaneous	Display the presenter's mouse motions when observing a shared Pro/ENGINEER session.	no, yes	yes

Name	Status	Category	Description	Values	Default Value
def_layer	Changed	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_assem_member, layer_assy_cut_feat, layer_axis, layer_chamfer_feat, layer_comp_design_model, layer_comp_fixture, layer_comp_workpiece, layer_copy_geom_feat, layer_corn_chamf_feat, layer_cosm_round_feat, layer_cosm_sketch, layer_csys, layer_curve, layer_curve_ent, layer_cut_feat, layer_datum, layer_datum_plane, layer_datum_point, layer_detail_item, layer_dgm_conn_comp, layer_dgm_highway, layer_dgm_rail, layer_dgm_wire, layer_dim, layer_draft_constr, layer_draft_dim, layer_draft_dtm, layer_draft_entity, layer_draft_feat, layer_draft_geom, layer_draft_grp, layer_draft_hidden, layer_draft_others, layer_draft_refdim, layer_driven_dim, layer_dwg_table, layer_ext_copy_geom_feat, layer_feature, layer_geom_feat, layer_gtol, layer_hole_feat, layer_intchg_func, layer_intchg_simp, layer_nogeom_feat, layer_nongraph_annot_elem, layer_note, layer_parameter_dim,	-

Name	Status	Category	Description	Values	Default Value
				layer_part_refdim, layer_point, layer_protrusion_feat, layer_quilt, layer_refdim, layer_ribbon_feat, layer_rib_feat, layer_round_feat, layer_sfin, layer_shell_feat, layer_skeleton_model, layer_slot_feat, layer_snap_line, layer_solid_geom, layer_surface, layer_symbol, layer_thread_feat, layer_trim_line_feat, layer_weld_feat	
default_layer_model	New	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_shrink_formula	New	Casting & Mold Design	Default option of shrinkage formula: 1/(1-S) - ASME standard or 1+S (simple).	asme, simple	simple
dim_offscreen_limit	Changed	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
dm_http_compression_level	Changed	Data Management	Set the data compression factor (0-9, 0=no compression) for data exchange with a Windchill server.	-	0
dm_overwrite_contents_on_update	New	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	Status	Category	Description	Values	Default Value
draw_models_read_only	Changed	Drawing	Yes - Drawing model files are read-only. Driven dimensions, gtols, and similar items cannot be added to the model. Cosmetic_only - Driven dimensions, gtols, cross sections, and explode states can be added to the model, design changes are not allowed.	yes, no, cosmetic_only	no
dwg_export_format	Changed	Data Exchange	Provides the ability to choose DWG file version when exporting from Pro/E drawing.	12, 13, 14, 2000	2000
dxf_export_format	Changed	Data Exchange	Provides the ability to choose DXF file version when exporting from Pro/E drawing.	12, 13, 14, 2000	2000
dxf_out_comments	Changed	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
ecad_other_outl_csyst_name	Changed	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_pin_hole_import	New	Data Exchange	Setup the default for the import of ECAD Holes from type PIN (IDF 3.0 ONLY)	yes, no	no
ecad_via_hole_import	New	Data Exchange	Setup the default for the import of ECAD Holes from type VIA (IDF 3.0 ONLY)	yes, no	yes
enable_face_preselection	New	Without Category		yes, no	yes
enable_implied_joints	Changed	Assembly	Allow underconstrained components as mechanism connections.	yes, no	yes
enable_k01_msr_anls	New	Without Category	Enable or disable new UI for Measure Analysis.	yes, no	no
enable_offset_fan_curve	New	Without Category	Enables fan curve option for offset tool.	yes, no	no

Name	Status	Category	Description	Values	Default Value
enable_protk_xml_schema	New	Application Programming Interfaces	Enables XML-Schema validation for Pro/TK feature XMLs	yes, no	no
expand_all_with_nested_layers	New	Without Category	Yes - expand all including nested sub layers.	yes, no	yes
fan_significant_locations_only	New	Electromechanical	Setup whether only significant locations should be flattened on flattening a harness.	no, yes	yes
fast_highlight	New	Model Display	Yes - Use simplified highlighting which redisplayes more quickly at the expense of certain details. This option may increase memory usage slightly.	no, yes	no
file_open_default_folder	Changed	File Storage & Retrieval	Sets default dir for File>Open. Working_directory - Search working directory. In_session - Search objects in session. Pro_library - Use only if Pro/LIBRARY is installed. Workspace - Use only with Pro/INTRALINK. Commonsense - Use only with Pro/INTRALINK.	default, working_directory, in_session, pro_library, my_documents	-
folder_display_file_limit	New	File Storage & Retrieval	Sets the default number of files that will be displayed when viewing the contents of a folder using the Folder Navigator.	-	-1
force_upd_assem_mp_in_simp_rep	Changed	Assembly	Update mass properties parameters even when some of the components of the assembly are not in master rep.	yes, no	no
general_undo_stack_limit	New	System	Setup the default number of entries in the top level Undo/Redo (default = 50).	-	50

Name	Status	Category	Description	Values	Default Value
iges_out_assembly_default_mode	Changed	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 geom info of comps and assy feats.	public flat, internal one level, internal all levels, public one_level, public all_levels, internal all parts, public all_parts	flat
instance_search_exhaustive	Changed	File Storage & Retrieval	Specifies extent of instance search. Yes - Include file type when entering instance name in File>Open: type in input panel (e.g. inst_name.prt) or select in drop-down list. If no .idx files are found, only working dir is searched; each generic is not.	no, yes	no
intf_in_dwg_view	Changed	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_granite_direct_enabled	Changed	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_use_variable_size	Changed	Data Exchange	Yes - IGES,DXF,STEP files with variable sized sheet information are imported and put on the appropriate variable-size format. Without a sheet size, the system tries to apply the correct variable-size format. No - The IGES,DXF,STEP drawing is placed on a standard sized format.	no, yes	yes

Name	Status	Category	Description	Values	Default Value
intf2d_in_acad_ignore_3d	New	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	New	Data Exchange	Create a single multi-line note on importing multi-line text when is set to YES.	yes, no	yes
intf2d_out_acad_mtext	New	Data Exchange	Create a single MTEXT entity for multi-line text notes when is set to YES.	yes, no	yes
intf3d_ideas_import_filter	New	Data Exchange	Enables UI to selectively import I-DEAS part and assembly models from the I-DEAS .mfl and .pkg container files.	yes, no	no
intf3d_ideas_install_dir	New	Data Exchange	Specifies the path for I-DEAS installation. Use the full pathname to avoid problems.	-	-
intf3d_ideas_run_command	New	Data Exchange	Specifies the I-DEAS run command. Default command is "ideas".	-	ideas.cmd
intf3d_parasolid_export_schema	New	Without Category		-	sch_10004
intf3d_ug_install_dir	New	Without Category	Specifies the path for Unigraphics installation. Use the full pathname to avoid problems.	-	-
isogen_nominal_size_map_file	Changed	Piping	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches	-	\$PTCSRC/text/piping_data/iso_data/isogen_nominal_size_map.pptd
jlink_java_command	New	Without Category	Specifies a command to start JRE for J-Link. May contain arguments and environment variables. Overrides the default, see also jlink_java2.	-	-

Name	Status	Category	Description	Values	Default Value
jlink_java2	Changed	Application Programming Interfaces	Option to force use of Java-2 JRE command line ("java" instead of "jre")	off, on	on
log_dwg_spr1150166_detection	New	Without Category	Create a log file and include if a particular drawing needs correction for SPR #1150166.	no, yes	no
mdo_cl_executable	New	Mechanism	Path for Custom Load executable.	-	-
mdo_enable_joint_friction	New	Mechanism	Enable joint axis friction.	yes, no	no
mdo_use_mm_body_order	New	Mechanism	Use Mechanica Motion body order. (Make the ground to be the first body in connection).	yes, no	no
mfg_auto_retract_clear_dist	New	Without Category		-	-0.001
mfg_process_print_dir	New	Without Category	Specifies the directory where Mfg Process html page get stored. Use the full path name to avoid problems.	-	-
mfg_process_table_setup_dir	New	Without Category	Specifies the directory where Mfg Process Table settings get stored. Use the full pathname to avoid problems.	-	-
mfg_process_template_dir	New	Without Category	Specifies the directory where Mfg Process Templates get stored. Use the full pathname to avoid problems.	-	-
mfg_start_model_dir	New	File Storage & Retrieval	Provides the complete path to the directory containing mfg start assemblies.	-	-

Name	Status	Category	Description	Values	Default Value
new_asm_regen_revnums	Changed	Assembly	Yes - Regenerate command will not increase the model revision number of the top assembly unless features belonging to the top assembly actually get regenerated. Regenerate command will increase model revision numbers of subassemblies if features in the subassembly regenerate. No - Regenerate command will always increase the revision number of the top assembly and almost never increase the revision numbers of subassemblies, regardless of whether or not assembly features regenerate.	yes, no	yes
number_user_colors	Changed	Colors	Specifies the maximum number of different wireframe colors that can display in the graphics area(s) at any given moment.	-	-1
orientation_style	New	Without Category	Set spin orientation style to dynamic or anchored.	anchored, dynamic	dynamic
overlays_enabled	Changed	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	yes
pdf_plot_config	New	Printing & Plotting	Set the pcf filename to be used when saving to PDF format.	-	-
pen1_line_weight	Changed	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	Status	Category	Description	Values	Default Value
pen2_line_weight	Changed	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	Changed	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	Changed	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	Changed	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	Changed	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	Changed	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	Changed	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
pro_unit_sys	Changed	Environment	Specifies the default system of units for new models.	mks, cgs, mmns, fps, ips, proe_def, mmks	-

Name	Status	Category	Description	Values	Default Value
read_parameters_of_excluded	New	Data Management	Allow Pro/ENGINEER designated parameters on excluded objects in Simplified Representation to be visible in Model Tree columns.	yes, no	no
relations_num_const_units	New	Without Category	Specifies whether diagnostic should be issued when an unitless numeric value causes relation expression evaluation trouble.	yes, no	no
relations_units_sensitive	New	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	New	Without Category	Yes - Sets the default of the ATTACH TYPE menu of the last options picked.	yes, no	yes
restricted_val_definition	New	Without Category	Specify name and path of the file contained restricted values definition.	-	-
reverse_spiral_finish_type	New	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.	-	0
save_hidden_items_status	New	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	Changed	File Storage & Retrieval	With family tables of solid parts, determines how instances are saved. None - Instance accelerator files not used. Explicit - Save instance accelerator files when instances explicitly saved. Always - Always save instance accelerator files.	none, explicit, always	none

Name	Status	Category	Description	Values	Default Value
save_model_display	Changed	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 View Performance dialog (shaded).	public wireframe, public shading_low, public shading_high, public shading_lod, internal no_display	shading_lod
show_all_mold_layout_buttons	Changed	Casting & Mold Design	Sets all mold layout buttons visible.	yes, no	no
sim_additive_mass	New	Simulation	Treats Mass Idealizations as additive if they are applied to the same entity and suppresses Mass precedence rules.	yes, no	no
sim_asm_modeling	Changed	Simulation	Controls automatic creation of assembly links and free connections in Mechanica. The default is YES.	yes, no	yes
sim_beamsection_path	New	FEM	Specifies the path where the Beam Sections will be stored.	-	-
sim_display_agem_controls	New	Simulation	Toggles display of AutoGEM Controls.	yes, no	yes
sim_display_fasteners	New	Simulation	Toggles display of Fastener icons.	yes, no	-
sim_display_measures	New	Simulation	Toggles display of Simulation Measure icons.	yes, no	yes
sim_display_mesh_controls	New	Simulation	Toggles display of Mesh Controls.	yes, no	yes
sim_display_mesh_quality	New	Simulation	Specifies the quality of mesh display in Mechanica. Values: FINE, MEDIUM, COARSE.	fine, medium, coarse	coarse
sim_display_mesh_shells_thick	New	Simulation	Toggles the display of shell thickness in Mechanica mesh. YES - Display shells with real thickness. NO - Display shells with zero thickness.	no, yes	no
sim_display_mesh_shrink_elems	Changed	Simulation	Controls display of Mesh through a specified percentage of shrinking of its elements.	-	-1

Name	Status	Category	Description	Values	Default Value
sim_html_report_prefs_file	New	FEM	Specifies the path to the Simulation post processor HTML report preferences file.	-	-
sim_motion_analysis_accuracy	Changed	Simulation	Specifies the default accuracy to which Mechanics determines values at each increment of the analysis. This number represents the acceptable fractional error in the computation of joint axis positions and rates at each time increment of the analysis. The default value is 0.0001.	-	0.0001
sim_motion_analysis_increment	Changed	Simulation	Specifies the default increment between steps at which Mechanics reports the motion of the mechanism. The default value is 0.1	-	0.1
sim_motion_analysis_integrator	Changed	Simulation	Controls the default integrator Mechanics uses when running the motion analysis. The value types are IMPLICIT (default) and EXPLICIT.	implicit, explicit	implicit
sim_motion_analysis_method	Changed	Simulation	Controls the default method Mechanics uses to analyze the motion of your model. The default is FASTEST_AVAILABLE.	fastest_available, time_integration, kinetostatic	fastest_available
sim_pp_path_absolute	New	Simulation	Choose Absolute (Yes) or Relative (No) path to "rwd" file.	yes, no	yes
sim_pp_template_dir	New	Simulation	Sets the default directory for results template files.	-	-
sim_pp_vector_plot_arrow_2d	New	Simulation	"Yes" plots 2D arrows, "NO" plots 3D arrows in Vector Plot of Mechanics Postprocessor.	yes, no	no
sim_run_out_dir	Changed	Simulation	Specifies the directory for Mechanics run output storage. By default, the current directory is used.	-	-
sim_run_tmp_dir	Changed	Simulation	Specifies the directory for Mechanics run temporary storage. By default, the current directory is used.	-	-

Name	Status	Category	Description	Values	Default Value
sim_smooth_aspect_ratio	New	Simulation	Sets the element aspect ratio criterion for the mesh smoothing process.	-	7
sim_smooth_edge_angle	New	Simulation	Sets the element edge angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	30
sim_smooth_skew	New	Simulation	Sets the element skew angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	45
sim_smooth_taper	New	Simulation	Sets the element taper criterion (Quad elements only) for the mesh smoothing process.	-	0.5
sim_smooth_warp_angle	New	Simulation	Sets the element warp angle criterion (Quad elements only), measured in degrees, for the mesh smoothing process.	-	10
sim_use_layers	Changed	Simulation	Allows layers to become groups for Mechanics Independent mode.	no, yes	no
simulation_fem_mode	Changed	Simulation	Yes - Activates FEM mode. No - Allows access to Mechanics only. Prompt (default) - Brings Mechanics 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	New	Simulation	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	New	Simulation	If set to yes, the Process Guide will be launched automatically upon entry into Mechanics, if template files are present.	yes, no	no
simulation_process_template_dir	New	Simulation	Specifies the default location of the Process Guide template files.	-	-

Name	Status	Category	Description	Values	Default Value
simulation_product	New	Simulation	Set Mechanica product being selected (Structure/Thermal/Motion). Prompt (default) - Brings Mechanica Model Type Dialog. This option is used only if information about Mechanica product is not yet stored with model.	public structure, public thermal, internal mold, internal motion, public prompt	prompt
sketcher_import_exact_geometry	Changed	Sketcher	During import in sketcher use exact geometry.	yes, no	yes
skip_small_surfaces	Changed	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
smt_mp_method	Changed	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
spherical_map_size	New	Drawing	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_rate_zoom_sensitivity	Changed	Model Display	Allows slower spinning when zoomed in on a model.	yes, no	no
spin_with_orientation_center	New	Without Category	Display orientation center while reorienting.	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.	public ap203_is, internal ap203_dis, public ap214_cd, public ap202_is, public 203_is_ext, public ap214_dis, public ap209_dis, public ap214_is	ap203_is

Name	Status	Category	Description	Values	Default Value
system_hidden_color	Changed	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
tbl_driven_tol_val_edit	New	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	Changed	File Storage & Retrieval	Specifies the designated template assembly. Use full path to avoid problems.	-	inlbs_asm_desi gn.asm
template_mfgcast	Changed	File Storage & Retrieval	Specify the model used as the default manufacturing cast template.	-	inlbs_mfg_cast. mfg
template_mfgcmm	New	File Storage & Retrieval	Specify the model used as the default manufacturing cmm template.	-	inlbs_mfg_cm m.mfg
template_mfgemo	New	File Storage & Retrieval	Specify the model used as the default manufacturing expert machinist template.	-	inlbs_mfg_emo .mfg
template_mfgmold	Changed	File Storage & Retrieval	Specify the model used as the default manufacturing mold template.	-	inlbs_mfg_mol d.mfg
template_mfgnc	New	File Storage & Retrieval	Specify the model used as the default manufacturing assembly template.	-	inlbs_mfg_nc. mfg
template_mold_layout	Changed	File Storage & Retrieval	Specify the mold layout assembly used as a default template.	-	inlbs_mold_lay .asm
triangulate_filled_areas	New	Drawing	Subdivide filled areas into triangles (may affect memory usage and plot files).	yes, no	no

Name	Status	Category	Description	Values	Default Value
ui_theme	New	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, standard	standard
verify_on_save_by_default	Changed	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
visible_message_lines	Changed	User Interface	Sets the default number of visible message lines in the Pro/ENGINEER message area.	-	6
web_browser_homepage	Changed	System	Enter location of Pro/E browser home page.	-	\$PTCSRC/html/usascii/pro/HELP/systop/resource_center_index.htm
weld_color	Changed	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