



## Pro/ENGINEER® Wildfire™ Configuration Options

The following table lists the configuration options for Pro/ENGINEER Wildfire. Descriptions and possible values for each option are provided. The first value listed is the default. For more information on setting and using configuration options, see the Pro/ENGINEER Help Center or click **Tools > Options** in Pro/ENGINEER.

Name	Description	Values
accuracy_lower_bound	Enter an accuracy value to override the default lower limit. The minimum value for relative accuracy is 1E-06.	
acis_out_version	Allows the user to choose ACIS export version.	4, 5
activate_sketch_with_dflt	Activate sketcher automatically using system defaults.	yes / no
add_java_class_path	Prepends the option value on the java environment variable, CLASSPATH, used to find classes in J-Link programs. To specify multiple search paths in the same line, delimit them using ':' on UNIX, and ';' on Windows NT.	
add_weld_mp	Yes - System includes welds when calculating mass properties. No - System excludes welds when calculating mass properties.	yes / no
aec_parameter_file	Specifies the file name of the user defined piping parameter codes.	
align_cable_bundles	Align NEWLY CREATED bundles where they meet or branch out	yes / no
allow_anatomic_features	Setting this configuration option to yes makes the following commands available when you click Insert > Advanced: Local Push, Radius Dome, Section Dome, Ear, Lip, Slot, Shaft, Flange, Neck.	yes / no
allow_confirm_window	Display a confirmation window when exiting Pro/ENGINEER. This enables you to use the mouse to confirm/cancel exiting Pro/ENGINEER.	yes / no
allow_float_opt_checkout	Yes - Allows checkout failure for floating modules.	no / yes
allow_move_attach_in_dtl_move	Yes - Move and Move Attach commands in drawing mode act together. No - Move and Move Attach commands in drawing mode do not act together.	yes / no
allow_move_view_with_move	Disallow the movement of drawing views with the mouse.	yes / no
allow_package_children	Sets what type of children packaged components can have. ALL - Features and components can be a children of packaged component. FEAT - Only features can be children of packaged components. NONE - Packaged components cannot have children.	all / feat / none
allow_ply_cross_section	Yes - Enables Pro/COMPOSITE to create a cross-section of composite plies.	yes / no
allow_redo_intersections	Yes - Displays the ReIntersect command in the ASSY FEAT menu.	yes / no
allow_ref_scope_change	No - Displays the message "Reference scope changes are prohibited by the configuration file settings" when the Ref Scope user interface is changed.	yes / no
allow_refs_to_geom_reps_in_drws	Yes - Allows you to create drawing references to geometry representations (includes dimensions, notes, and leaders). These references may become invalid if the referenced geometry changes resulting in geometry representations not updating in drawings.	yes / no
allow_rfs_default_gtols_always	Yes - Enables RFS/Default gtols to be created even when not permitted by ANSI standard.	no / yes
allow_save_as_instance	Allows the option to save a copy of the active model as a family table instance.	yes / no

Name	Description	Values
allow_sketch_selection	Yes - An external sketch may be selected as the feature's section. The sketch is copied into the feature without an associative link. No - An external sketch may not be selected as the feature's section.	yes / no
allow_udf_style_cosm_threads	Defines the Cosmetic Thread user interface. Yes - Creates a Cosmetic Thread as a UDF. No - Creates a Cosmetic Thread as a Cosmetic Thread feature.	yes / no
allow_workpiece_silhouette	Determines if the workpiece is allowed for Silhouette Mach Window creation. NO - only reference part is allowed; YES - both workpiece and reference part are allowed; AUTO - only workpiece is allowed.	yes / no / auto
ang_dim_in_screen	Yes - If the display is in the default pan/zoom setting (View>Pan/Zoom>Reset), the system checks whether an angular dimension is visible on the screen. If it is not visible, then the system moves the dimension to a visible location.	yes / no
ang_units	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
angle_grid_interval	Modify grid space in angular units for handle movement.	
angular_tol	An alternative format for setting default angular tolerance dimensions. The value sets the number of decimal places, and the tolerance is the actual tolerance value. For integer dimensions, the value is zero, and the tolerance is an integer.	
angular_tol_0.0	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.	
angular_tol_0.00	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.	
angular_tol_0.000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.	
angular_tol_0.0000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.	
angular_tol_0.00000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.	
angular_tol_0.000000	Sets the default tolerance for angular dimensions. Each option sets a tolerance for a specific decimal place. The value is the place of the last decimal.	
arc_radius_limit	Sets the upper limit for a radius in a CIRCLE statement of NCL file. The value must be positive. If not, 10*part_size is used.	
assy_mfg_open_mode	No Default Application set for opening Assembly NC Models.	none / mfg / process / feature / toolmaker
atb_auto_check_on_activate	Sets state of ATB 'Auto Check Status On Activate' toggle.	on / off
atb_auto_check_on_retrieve	Sets state of ATB 'Auto Check Status On Retrieve' toggle.	on / off
atb_auto_check_on_update	Sets state of ATB 'Auto Check Status On Update' toggle.	on / off
atb_ident_cadds_files	Allows to identify CADDs files in the file open browser.	yes / no
atb_search_path	Extends search path for ATB Update and ATB Check Status commands.	
atb_show_log	Sets state of ATB 'Show Log' toggle.	on / off
attach_menumanager	Setup menu manager attachment to the main Pro/E window.	yes / no
auto_assembly_with_layouts	Yes - Enables automatic assembly. No - Disables automatic assembly.	yes / no

Name	Description	Values
auto_associate_dimensions	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
auto_constr_always_use_offset	Control whether auto constraint should create offsets. Yes - Auto constraint always creates offsets. No - Auto constraint snaps align or mate if surfaces are within tolerance.	yes / no / never
auto_constr_offset_tolerance	Sets the auto constraint tolerance for creating offset dimensions. If distance is less than this tolerance multiplied by component size, offset is set as coincident. Default value is 0.5 .	
auto_convert_cables	Yes - When regenerating a pre-Release 11.0 cabling assembly, the system automatically creates a separate spool feature for every individual wire or cable in the assembly. No - An interface for the cable conversion appears.	yes / no
auto_regen_views	Yes - Automatically repaints the drawing display when changing from one window to another. No - Update the drawing only. Click View>Regen View>Pick View>Current Sheet or All Sheets. Neither Repaint nor Regenerate update the drawing.	yes / no
auto_xml_on_retrieve	Automatically load Cabling Logical Reference from an XML file when retrieving the cabling assembly	yes / no
auto_xml_on_save	Automatically create an XML Logical Reference when saving a cabling assembly	yes / no
autodrill_udf_dir	Storage directory for holmaking MUDF's that are used by Pro/NC Auto-drilling functionality	
autodrilling_config_filename	Retrieve automatically and load the configuration file for the auto-drilling table.	
autodrilling_holestyle_param	Parameter name included in hole features of a manufacturing reference model. The parameter value identifies the MUDF for holmaking that is automatically applied in Pro/NC to drill these holes.	
autoplace_single_comp	Assembling components with an interface will automatically place the component at the first position satisfying the interface definitions.	no / yes
autoroute_path_param_name	Set the wire parameter to use to select a path during autorouting.	
bell	Yes - Turns on the keyboard bell that rings after each prompt. No - Turns off the keyboard bell. To override this setting use the Environment dialog box on the Tools menu.	yes / no
bitmap_size	Works in conjunction with save_bitmap. Determines the size (squared) of the bitmap image that is stored.	
blank_layer	Blanks specified layers when you begin a Pro/ENGINEER session. The value is the layer ID.	
blended_transparency	Yes - Transparent colors will appear using alpha blending (if supported) when the model is shaded.	no / yes
bm_graph_tool	BMX graph tool	default / generic_graphtool
bmgr_pref_file	Specifies the location of the Graph Tool preference file.	
bom_format	Sets the BOM format file to be used for a customized BOM. Specify name and path.	
browser_favorite	Specifies the directories visible in the file browser for quick navigation. Use the full path to avoid problems.	
bump_revnum_on_retr_regen	Determines whether or not revnum 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 revnum bumping).	yes / no

Name	Description	Values
button_name_in_help	Yes - The name and menu of any selected menu option displays, in English, in the help text associated with that button.	yes / no
cable_int_portions_for_clr	No - Global clearance checking for internal cable portions is excluded. Yes - Global clearance checking for internal cable portions is included.	no / yes
cadam_line_weights	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).	
cadds_import_layer	Allows import of CADD5 layering.	yes / no
campost_dir	Specifies the CAM-Post load point directory to enable you to directly set-up the post-processors and generate MCD files.	
can_snap_to_missing_ref	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
capped_clip	Yes - Displays the model as a solid when shaded and clipped. No - Displays the model as surfaces when shaded and clipped.	no / yes
catia_out_to_existing_model	Append - If the selected CATIA model already exists, the new data is appended to the existing CATIA file. Overwrite - If the selected CATIA model already exists, the newly exported file overwrites the existing one.	append / overwrite
catia_translator	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
cdt_transfer_details	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
centimeter_grid_interval	Modify grid space in centimeter units for handle movement.	
cgm_inc_pad_byte_in_length	Yes - Enables a metafile to be processed by the Micrographic CGM converter.	yes / no
cgm_use_enum_in_real_spec	Yes - Enables a metafile to be viewed in Advanced Technology Center's ForReview.	yes / no
cgm_use_reversed_ieee_floats	Yes - Enables a metafile to be viewed in Advanced Technology Center's ForReview.	yes / no
chamfer_45deg_dim_text	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
check_intralink_attributes	Specify whether intralink attributes should be checked or not.	yes / no
check_same_assy_parts	Determines whether to permit different names for different part occurrences, or not in mirror subassembly UI.	yes / no
chk_part_surfs_profpack	Yes - Includes all reference part surfaces as check surfaces for profiling and pocketing NC sequences.	yes / no
cl_arrow_scale	Enables you to control the size of the tool path arrow for contouring NC sequences. 0 - The arrow does not display. Any other value - The arrow scales accordingly.	
cl_rotate_sign_convention	TOOL or TABLE - defines ROTATE sign direction convention based with respect to either rotation table or cutting tool.	table / tool
clearance_triangulation	Specifies the quality of surface triangulation used to obtain initial guess points for clearance and interference calculations.	none / low / medium / high
clip_always	Yes - Display clipping after Visibilities dialog box is closed.	no / yes

Name	Description	Values
clock	Yes - Displays the clock while Pro/ENGINEER is working.	yes / no
clr_print_plus_minus	Sets the display of the system accuracy when calculating clearances between two parts or surfaces. The default is to not display the accuracy.	yes / no
cmdmgr_trail_output	Trail file and mapkey output will include system instructions that are independent of user interface configuration when available (typically menu bar and toolbar options).	yes / no
color	Turns the colors on or off. Turning colors off displays the model wireframe in white.	no / yes
color_ramp_size	Specifies the number of shades in a color ramp. Applicable to multi-color, shaded model displays of simulation / analysis results. System graphics must support 256 colors and color maps that compress.	
color_resolution	Specifies minimum allowable deviation between user-defined colors. Colors having RGB values within tolerance of existing colors cannot be created. Decreasing this setting allows more colors that are very similar in RGB value to be defined.	
color_windows	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
comp_angle_offset_eps	Specifies the angle epsilon such that if the desired surfaces are equal or greater than the epsilon (in degrees), than an angle offset constraint will be created.	
comp_assemble_start	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
comp_offset_dec_places	Sets the default number of decimal places (0-10) to display in the offset column of component placement.	
comp_rollback_on_redef	No - Assembly is not rolled back when the user redefines a component.	yes / no
comp_snap_angle_tolerance	Specify the angle tolerance for snapping while freeform dragging components. Default value is 30.0 degrees.	
comp_snap_dist_tolerance	Specify the distance tolerance for snapping while freeform dragging components. Default value is 0.1 (relative to size of component being assembled).	
company_name	Used with the export option iges_out_mil_d_28000. To support MIL-D-28000, your company name is required input and the system prompts you every time you export through IGES unless you set the company name in your configuration file.	
compress_output_files	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
conf_mouse_anim	Display the presenter's mouse motions when observing a shared Pro/ENGINEER session.	no / yes
copy_dxf_dim_pict	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
copy_geom_update_pre_2000i_dep	Yes - Flags independent copy geom features in a pre-2000i model as modified when retrieved into Pro/ENGINEER. Save the model immediately to update the model's copy geom dependency information.	no / yes
cpus_to_use	Set up the number of threads including main thread	

Name	Description	Values
create_drawing_dims_only	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
create_fraction_dim	Yes - All dimensions created will be displayed as fractions.	yes / no
create_numbered_layers	Yes - Creates default layers named 1 through 32.	yes / no
create_temp_interfaces	Allow the automatic creation of interfaces based on previous assembly instructions.	yes / no
curr_proc_comp_def_color	Specifies the default color used to display the current component in a process assembly.	
curr_proc_comp_def_font	Specifies the default font used on the current component in a process assembly.	
custom_rainbow	Used to specify fringe colors in Pro/FEM-POST results. Specify blue, black, cyan, magenta, green, yellow, red, and white. Enter colors as a dash-delimited string (lowest to highest value). The default is blue-cyan-green-yellow-red-magenta-white.	
datum_point_symbol	Specifies the shape of datum point symbols displayed in Part or Assembly mode.	dot / cross / circle / triangle / square
dazix_default_placement_unit	Specifies the units to be used for data imported in Dazix files.	micron / mm / thou
dazix_export_mounthole	Yes - Causes the MOUNTHOLE section of a Dazix file to be processed as a mount hole. No - Causes the MOUNTHOLE section to be processed as a cut.	yes / no
dazix_z_translation	Yes - Passes the objects in the .edn files through z translation.	yes / no
def_layer	Specifies default layer names for different types of items. The first value string is the layer type. The second value string is the layer name.	
default_abs_accuracy	Defines the default absolute part accuracy.	
default_ang_dec_places	Specifies the number of decimal places shown in angular dimensions in a drawing.	
default_comp_geom_scope	Set the specified Shared Geometry reference control conditions to newly added components.	all / pg / none
default_comp_invalid_refs	Set the specified handling method of out-of-scope Accessible references to newly added components.	copy / prohibit
default_comp_scope_setting	Set the specified Accessible reference control conditions to newly added components.	all / none / subassemblies / skeleton_model
default_comp_skel_geom_scope	Set the specified Shared Geometry reference control conditions to newly added Skeleton components.	all / pg / none
default_comp_skel_scope_setting	Set the specified Accessible reference control conditions to newly added Skeleton components.	all / none / subassemblies / skeleton_model
default_dec_places	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.	
default_dim_num_digits_changes	Sets the default number of digits displayed in a dimension to the last entered value. No - The system defaults to the value specified for the configuration file option default_dec_places.	yes / no
default_draw_scale	Sets the default drawing scale for views added with the No Scale command. The value must be greater than 0. No - The system does not set a default drawing scale.	
default_ext_ref_scope	Set default scope for externally referenced models. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all / none / subassemblies / skeleton_model



Name	Description	Values
default_font	Sets font for text other than menu bar, menus and their children, and pop-up menus and help. Add comma-delimited variables in any order (italic bold, 24, times or 24, times, italic bold have the same effect). Any omitted variable uses standard setting.	
default_geom_scope	Default value for Geometry Scope allowed for referencing.	all / pg_if_exists / pg_only
default_mold_base_vendor	Default Value for Mold Base vendor.	futaba_mm / dme / hasco / dme_mm / hasco_mm
default_obj_constraints_scope	Set the specified Shared Placement reference control conditions to newly created models if template models were not used.	all / interfaces / none
default_object_geom_scope	Set the specified Shared Geometry reference control conditions to newly created models if template models were not used.	all / pg / none
default_object_invalid_refs	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
default_object_scope_setting	Set default condition for reference control. All - Any model. None - Only current model and children. Skeletons - Any component in model's assembly and higher skeletons on branch. Subassembly - Only components and children in model's assembly.	all / none / subassemblies / skeleton_model
default_placement_scope	Default value for Component Placement Reference filter.	all / intrf_if_exists / intrf_only
default_ramp_size	Specifies a number of shades of gray to show shading of non-colored objects.	
delete_after_plotting	Yes - A plot file is automatically deleted from the directory in which it resides when it has been successfully plotted.	yes / no
depthcue_always	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
depthcue_value	Aids in visualizing a wireframe model. With depth cue enabled, this option specifies the default percentage of depthcue used. Depth cue is supported only by the appropriate graphics hardware.	
diagram_pin_edit_multi_line	(Diagram) Determines the Pro/TABLE format used when you are modifying pin parameters. "yes" - You can add user-defined pin parameters by including them between the DEFINE and ENDDEF statements for each pin. "no" - The system uses columnar format and	yes / no
dialog_translation	Specifies the language in which the dialog display when running non-English version of Pro/ENGINEER.	yes / no
dim_fraction_denominator	Sets the largest denominator to be used for fractional dimensions. If the fraction can be reduced, then it converts to the lowest possible denominator (for example, 4/32 converts to 1/8).	
dim_fraction_format	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
dim_offscreen_limit	If more than zero, sets the margin beyond which dimensions are automatically snapped back onto the screen. For example, a value of .2 lets a dimension be 20% of the window size outside the window. If set to zero, disables snapping of offscreen dimensions.	
dir_cache_max	Specify the maximum number of directories to cache. Larger values can speed up file access, but use more memory.	

Name	Description	Values
direct_vrml	Yes - Enables direct VRML conversion.	yes / no
disable_search_path_check	Controls whether the search path is checked for name conflicts when creating, renaming, or copying models. Name conflicts should be avoided because different models with the same name cannot be together in session.	yes / no
disp_in_open_rep_create	Simplified Rep to display initially in Open Rep Create: "No" for none, "Yes" for graphics rep, or "Default Rep" to display the rep indicated by option "open_simplified_rep_by_default".	yes / default rep / no
disp_trimetric_dwg_mode_view	Displays the model in default orientation when placing a general view on the drawing. No - Model does not appear until Default is chosen from the Orientation dialog box.	yes / no
display	Wireframe - Displays model with all lines in white. Hiddenvis -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 / hiddenvis / hiddeninvis / shade
display_axes	Displays or hides datum axes.	yes / no
display_axis_tags	Displays or hides datum axis name tags.	yes / no
display_comps_to_assemble	Yes - The design model goes into memory and displays. Pick process components from the design model or the Model Tree. No - Only the Model Tree displays. As you pick process components from the Model Tree, they go into memory.	yes / no
display_coord_sys	Yes - Coordinate systems will be displayed. No - Coordinate systems will not be displayed.	yes / no
display_coord_sys_tags	Yes - Coordinate system name tags will be displayed. No - Coordinate system name tags will not be displayed.	yes / no
display_dwg_sketch_constraint	When set to yes, parametric sketching constraints, such as V for vertical, are displayed when a drawing object is selected	yes / no
display_dwg_tol_tags	Sets the display of the tolerance tags that appear in drawings. This option does not affect the display of tolerances on dimensions.	yes / no
display_florin	Yes: display of the florin (international currency symbol) where possible. No: use the euro symbol exclusively.	no / yes
display_full_object_path	Controls the display of the object's full file name (including its object-type suffix and its version number) and file path in the window title and Model Info display. Yes - Displays full file name and file path. No - Displays object name only.	no / yes
display_in_adding_view	Controls model display in a drawing when auto_regen_views is No. Wireframe - Adds new views in wireframe, displays datums. Minimal_wireframe - Displays in wireframe, no datums, axes, or silhouette edges. Default - Displays using the Environment setting.	wireframe / minimal_wireframe / default
display_internal_cable_portion	Yes - Internal cable portions will be displayed for any wires that have been manually set to Yes in the Cable Paths Setup dialog box for each individual splice and custom component. This option can be overridden in the Environment dialog box.	yes / no
display_layer	This causes the specified layers to display when you begin a Pro/ENGINEER session. The variable "value" is the layer Id.	
display_plane_tags	Yes - Datum plane name tags will be displayed. No - Datum plane name tags will not be displayed.	yes / no
display_planes	Yes - Datum planes will be displayed. No - Datum planes will not be displayed.	yes / no
display_point_tags	Yes - Datum point tags will be displayed. No - Datum point tags will not be displayed.	yes / no
display_points	Yes - Datum points and their names will be displayed. No - Datum points and their names will not be displayed.	yes / no



Name	Description	Values
display_silhouette_edges	Sets the display of silhouette edges for wireframe display only.	yes / no
display_thick_cables	Yes - Display cables and wires with thickness. No - Display the centerline of cables and wires only. This option can be overridden in the Environment dialog box.	yes / no
display_tol_by_1000	If set to Yes, tolerances for non-angular dimensions will be displayed multiplied by 1000.	yes / no
dm_cache_mode	Indicate which objects will be written to local cache when objects in Pro/E memory are saved.	none / modified / all
dm_cache_size	Enter amount of disk space (in megabytes) to allocate for local file storage.	
dm_http_compression_level	ND files will be compressed before being sent for upload by setting the value of config.pro option 'dm_http_compression_level' to a value between 1 and 9. If the value is not set or a 0-value is set, it means no-compression. This change can speed up the Upload operation for a client connected to a Windchill server over a slow network.	
dm_remember_server	Primary server/workspace must be set by user for each Pro/E session.	no / yes
dm_secondary_upload	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
dm_upload_objects	Indicate when modified Pro/E objects will be moved from the local cache to the user workspace on the server.	automatic / explicit
dont_show_symbol_nodes	Do not show references derived from relations in Parent/Child viewer.	yes / no
draw_models_read_only	Yes - Drawing model files are read-only (i.e. they cannot be modified). You cannot add driven dimensions, geometric tolerances, or similar items to the models.	yes / no
draw_points_in_model_units	Defines the current draft view's coordinate values as model units rather than drawing units. The GET POINT menu uses the scale of the draft view and the draft view's model units for relative and absolute coordinate entry and display in the Message Window.	yes / no
drawing_file_editor	Specifies the default 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
drawing_setup_file	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.	
drawing_view_origin_csys	The named coordinate system will be used as the origin of a newly created view or of a modified view. None - The system will not use a previously specified coordinate system.	none /
dwg_export_format	Provides the ability to choose DWG file version when exporting from Pro/E drawing.	12 / 13 / 14
dxf_block_to_pro_symbol	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
dxf_export_format	Provides the ability to choose DXF file version when exporting from Pro/E drawing.	12 / 13 / 14
dxf_export_mapping_file	Specifies the DXF export mapping file. Absolute or relative path to the file can be used.	
dxf_in_faceted_brep	Default, faceted boundary representation is allowed in import.	yes / no
dxf_out_comments	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
dxf_out_drawing_scale	Specify whether or not to export drawing scale to the DXF or DWG file. Yes - Includes a scale factor DIMLFAC in the exported DXF/DWG. No - Exports without using a scale for the drawing. Set to Yes when exporting to read into AutoCAD.	yes / no

Name	Description	Values
dxs_out_scale_views	Specify whether or not to scale drawing views when exporting to the DXF or DWG file. Yes - rescales the entire drawing in such a way that the main view scale becomes 1:1. No - exports without rescaling.	yes / no
dxs_out_sep_dim_w_breaks	YES - Breaks dimensions with breaks on witness lines to separate entities and preserves the original picture. No - Exports such dimensions as DXF DIMENSION entities and removes the breaks on witness lines.	yes / no
dxs_out_stroke_text	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
dxsio_in_sjis	Determines if the SJIS is supported when importing/exporting DXF files in Japanese	yes / no
ecad_area_default_import	Defines how imported ECAD areas are treated. Cosm_area - Treats imported ECAD areas as cosmetic area features. 3d_volume - Imports ECAD areas with Z-heights as a 3D enclosed quilt.	3d_volume / cosm_area
ecad_board_csys_def_name	Specifies the default coordinate system name added to an ECAD board being imported. If you do not set this variable, the system prompts you for a name.	
ecad_comp_csys_def_name	Specifies the default coordinate system name added to an ECAD component being imported. If you do not set this variable, the system prompts you for a name.	
ecad_comp_layer_map	Allow layer mapping for ECAD component's import into Assembly.	yes / no
ecad_create_hint_add	Assists in creation of an ecad_hint.map file. Yes - Automatically renames components, if necessary, each time library of component outlines is imported to Pro/ENGINEER. This does not create an ecad_hint.map file. It controls if ecad_hint.add is created.	no / yes
ecad_default_comp_height	Sets default value and units for an ECAD component being imported. Units can be: inch, mil (1E-3 inches), thou (1E-6 inches), cm, mm, micron (1E-6 meters), dsu (1E-8 meters). If not set, the system uses the current component's units.	
ecad_default_comp_place_status	Sets default component placement status for export (parameter setting will override the default).	unplaced / fixed / placed / mcad / ecad
ecad_exp_both_two_areas	Supports the export of ECAD areas with different "Above Board" and "Below Board" conditions. "yes" - Enables you to export both sided keep-in/keep-out ECAD areas as two individual areas (top and bottom).	yes / no
ecad_export_holes_as_cuts	Exports Pro/ENGINEER holes as cuts to ECAD systems.	yes / no
ecad_import_holes_as_features	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
ecad_mapping_file	Specify the ecad_hint.map file that will be used for ECAD operations.	
ecad_missing_component_status	Setup default status for missing components in "Investigate Placement" Dialog.	keep_missing / delete_missing
ecad_other_outl_csys_def_name	Specifies the default coordinate system name for the .OTHER_OUTLINE section of an IDF 2.0 ECAD component being imported. If you do set this variable, the system prompts you for a name.	
ecad_panel_csys_def_name	Specifies the default coordinate system name added to an ECAD panel being imported. If you do not set this variable, the system prompts you for a name.	
edge_display_quality	Controls display quality of an edge for wireframe and hidden-line removal. Normal - Normal quality of edge display. High - Increases tessellation by 2. Very_high - Increases tessellation by 3. Low - Decreases tessellation from normal, speeds up display.	normal / high / very_high / low

Name	Description	Values
enable_absolute_accuracy	Controls display of the ACCURACY menu. Yes - ACCURACY menu always appears when you choose Accuracy from the PART SETUP menu. No - The menu appears only if the part is currently defined with absolute accuracy.	yes / no
enable_assembly_accuracy	No - Disables accuracy modification for assembly objects.	yes / no
enable_cadra_export	Enables the CADRA command in the EXPORT menu. This enables the creation of a CADRA specific IGES file.	no / yes
enable_hlr_hashing	Enables hlr hashing	no / yes
enable_popup_help	Controls pop-up help in dialog boxes. Yes - Help enabled. No - Help disabled.	yes / no
enable_tree_indep	Setup whether tree customization will allow separating the tree window from Pro/E window.	yes / no
erv_show_external_only	Yes - Global Reference Viewer will show objects with external references only; No - all the objects will be visible	yes / no
excel_analysis_directory	Sets the default directory for the MS Excel files to be used for Excel analysis calculations. Use the full pathname to avoid problems.	
exec_prog_on_assem_retr	Execute pro/program for models during assembly retrieval.	yes / no
explode_iges_dimension_note	Controls how dimensions are treated when importing an IGES drawing file. Yes - Each IGES dimension explodes into an independent note with the dimension text and a dimension with an empty note. No - Dimensions are treated as before.	yes / no
export_3d_force_default_naming	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
export_to_shipit	Enable Ship-it interface export.	yes / no
extend_cvpath	Extends CVPATH search path for CADD5 data exchanging.	
fail_ref_copy_when_missing_orig	Yes - When using a copied geom feature or a backed-up external reference with the Design Manager, and the reference is missing in the original, then the copied geom feature fails. No - The feature freezes and does not update.	yes / no
fail_unattached_feature	Handles unattached protrusion and cuts, when retrieving pre-Release 15 models. Yes - When a part is retrieved, the system fails an unattached protrusion or cut. No - When a part is retrieved, the system does not fail an unattached protrusion or cut.	yes / no
fan_with_network	Setup whether the network should be included on flattening a harness.	yes / no
fasthlr	Specifies whether fast hidden line removal is on/off by default.	no / yes
feature_create_auto_begin	In feature creation, determines whether or not the Define button is automatically activated when you open the dialog box. Yes - Button is activated automatically. No - You select the button.	yes / no
feature_create_auto_ok	In feature creation, determines whether or not the system activates the dialog box OK button automatically when the last required element is defined. Yes - Button is activated automatically. No - You select the button.	yes / no
fem_allow_named_mesh_files	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
fem_ansys_annotations	Toggles transfer of loadcase names to ANSYS.	yes / no
fem_ansys_grouping	Toggles transfer of groups/layers to ANSYS.	yes / no
fem_asp_ratio	The value that FEM created elements of aspect ratio are compared against.	
fem_default_solver	Specifies default solver.	ansys / nastran

Name	Description	Values
fem_dist_index	The value that FEM created elements of distortion indices are compared against.	
fem_edge_angle	Sets the minimum acceptable default angle between adjacent element edges. Enter value between 0 and 90.	
fem_ignore_unpaired	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
fem_measures	Provides access to Simulation Measures in FEM mode.	yes / no
fem_mesh_operations	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
fem_mesh_preserve	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
fem_mid_ratio	The value that FEM created elements of mid ratios are compared against.	
fem_neutral_version	Specifies version number of neutral file.	
fem_new_tetra_mesher	Yes (default) - Use new FEM Tetrahedral Mesher. No - Use original FEM Tetrahedral Mesher.	yes / no
fem_remove_unopposed	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
fem_skew_angle	Sets the maximum acceptable default skew angle value, measured in degrees. Enter a value between 0 and 90.	
fem_solid_shell_auto_constraint	Yes - Rotational constraints will be created at the interfacing nodes between solids and shells. No - The user needs to make sure that the interfacing nodes are not underconstrained (like adding beams). Ask - Prompt the user before running.	yes / no / ask
fem_solver_time_limit	For NASTRAN solution, uses this option to interrupt the solver after the specified time limit. The default value is 60 minutes.	
fem_taper	Sets the minimum acceptable default taper value. Enter a value between 0 and 1.	
fem_twist_angle	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.	
fem_warp_angle	Sets the maximum acceptable default warp angle value, measured in degrees. Enter a value between 0 and 90.	
fem_which_ansys_solver	Specifies the ANSYS solver to be used.	frontal / iterative / powersolver
fem_y_dir_compress	Specifies the Y-direction based on compressed and uncompressed geometry. The default value is uncompressed.	yes / no
file_open_default_folder	Sets default dir for File>Open. Working_directory - Search working directory. In_session - Search objects in session. Pro_library - Use only if Pro/LIBRARY is installed. Workspace - Use only with Pro/INTRALINK. Commonspace - Use only with Pro/INTRALINK.	default / working_directory / in_session / pro_library
fix_autocad_iges_text_scale	Fixes scaling problems with AutoCAD releases earlier than Release 10. Yes - Repairs AutoCAD files appropriately.	yes / no
fix_boundaries_on_import	Controls fixing of boundaries of imported surfaces. Yes - Fixes boundaries by zipping gaps and correcting tangencies. No - Does not fix boundaries.	yes / no

Name	Description	Values
fix_catia_iges_sym_note	Corrects a problem caused by multiple rotations of a note in a CATIA IGES file. Yes - Rectifies any discrepancies.	yes / no
fix_imported_set_view_orient	Sets viewing characteristics for imported CATIA SET files. Yes - Files can have translated views. No - Files cannot have translated views.	yes / no
fix_refs_to_intersections	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
flip_arrow_scale	Sets a scale factor for enlarging the size of the flip arrow that appears for feature creation direction (default = 1).	
fonts_size	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	Modify grid space in foot units for handle movement.	
force_new_file_options_dialog	Force the use of the New File Options dialog.	yes / no
force_wireframe_in_drawings	Yes - Displays all views of drawings in wireframe. No - Displays views according to the display set for the session (wireframe, hidden, and so on).	yes / no
format_setup_file	Assigns a specified setup file to each drawing format. To assign the drawing's parameter values to the format, you must retrieve the drawing's setup file into the format.	
frames_per_sec	Determines whether view is animated.	no / yes
frames_per_sec_value	Used to set the smoothness of view animation changes. Higher numbers provide more frames/sec, and giving you a smoother animation. Lower numbers cause the view to change more quickly.	
freeze_failed_assy_comp	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
full_hlr_for_cables	Full - Removes hidden lines from view when cables hide other geometry (use when Hidden Line active). Partial - Cables hide other non-interfering cables except when cables route together between same locations. None - Display hidden lines (faster).	none / full / partial
gerber_360_interpolation	Yes - Arcs and circles included in outputted Gerber plot files will take this format.	no / yes
gpostpp_dir	Specifies the directory containing the post processor files to be used by Pro/NCPOST (GPOST).	
graphics	Sets the optional graphics environment used by certain platforms for running Pro/ENGINEER. Refer to the Installation Guide for a list of defaults and optional graphics.	opengl / x_windows / starbase / xgl / no_graphics
grid_snap	Yes - Makes pick points snap to a grid. No - Turns the grid snap off so that any location can be picked.	yes / no
gtol_dim_placement	Controls display of geometric tolerances in Part. Use like drawing set-up option of the same name. On_bottom - Gtols display at bottom (below additional lines of text). Under_value - Gtols display under dimension value (above additional lines of text).	on_bottom / under_value
harn_tang_line_display	Display the tangency lines between cable segments in thick cable display mode.	yes / no
help_translation	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
highlight_erased_dwg_views	Controls the display of erased view outlines (environment option Highlight Erased Views). Yes - Highlights erased views. No - Turns off highlighting for erased views.	yes / no

Name	Description	Values
highlight_layer_items	Specifies whether items selected in the layer tree should be highlighted in the graphics area.	yes / no
highlight_new_dims	In Drawing mode, highlights new dimensions in red until you move them or repaint the screen.	yes / no
hlr_for_quilts	In Drawing, controls quilt display in hidden line removal. Yes - Includes quilts in the hidden line removal process (preselects Qlt HLR in the VIEW DISP menu). Default in the VIEW DISP menu, uses the value you have set.	yes / no
hole_diameter_override	yes — In the Hole dashboard, you can change the diameter of a suggested default diameter for standard tapped and clearance holes. Set this to yes if you are familiar with available drills and need to change the system default value. no - Pro/ENGINEER displays a diameter value (based on the table lookup function) and grays out the value so you cannot change it.	yes / no
hole_parameter_file_path	Enables you to use a local hole table instead of the system hole table.	
iconify_entire_pro	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
iges_clip_view_ent	Determines whether IGES entities are clipped with regards to IGES views. YES - Entities outside the view outline are clipped. NO - No clipping.	yes / no
iges_clip_view_note	Determines how IGES notes are clipped with regards to IGES views. No_clip - No clipping. Full_clip - Notes that are partially outside the view outline are clipped. Partial_clip - Notes completely outside the view outline are clipped.	no_clip / full_clip / partial_clip
iges_export_dwg_views	Yes - Exports drawing view information. No - Does not export drawing view information.	yes / no
iges_in_106_f2_as_spline	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
iges_in_assoc_dim_geom_21	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
iges_in_dwg_color	Yes - Import RGB information to IGES files. The color definition entities in the IGES file are imported to the drawings as user-defined colors. No - The color-definition entities in the IGES files are ignored when importing to the drawing.	yes / no
iges_in_dwg_line_font	Controls the import of user-defined line fonts. Yes - Give default names to user-defined line styles without names in order ("IGES_1," "IGES_2" etc.). No - Imports the user-defined line fonts as Solid line font.	yes / no
iges_in_group_to_dwg_layer	Yes - Converts an IGES group to drawing layers. No - The IGES group is not converted.	no / yes
iges_note_disp	Determines what to do when IGES note entities do not point to any views or the drawing entity. See the configuration option iges_zero_view_disp.	as_geometry / all_views / no_views / as_is
iges_out_all_srfs_as	Default - Outputs all surfaces as appropriate IGES surfaces. 114 - This option applies only to representations of surface shape. Trimmed surface entities (type 144) are exported regardless of setting. 128 - Outputs all surfaces as IGES B-spline surfaces.	128 / 114 / default
iges_out_assembly_default_mode	Set default for assembly export via IGES. Flat - Single level model. One_level - Assy structure with pointers to components. All_levels - Assy structure with all comps to IGES files. All_parts - Assy to IGES files with geom info of comps and assy feats.	flat / one_level / all_levels / one_level / all_levels / all_parts / all_parts



Name	Description	Values
iges_out_catia_gdt_width	Yes - Enables the desired width of a gtol symbol to be exported to CATIA.	yes / no
iges_out_catia_notes	Yes - Breaks a large note (greater than 70 strings) down into smaller notes, each producing an IGES entity.	yes / no
iges_out_dwg_color	Yes - Enables the export of RGB information to IGES files. The user-defined colors in the drawing are exported to an IGES file as color definition entities. No - Ignores the user-defined colors in the drawing when exporting IGES file.	yes / no
iges_out_dwg_line_font	Controls the export of user-defined line font through IGES. No - Exports all geometry as solid font.	yes / no
iges_out_ent_as_bspline	True - Exports all geometry entities, other than lines or arcs, as third-degree B-splines. False - Does not export entities as third-degree B-splines.	true / false
iges_out_jamais_compliant	Yes - Specifies IGES output with special JAMA-IS subset specification, compliant with the version 1.02 of JAMA-IS (Japan Automobile Manufacturers Association IGES Subset Specification). No - Specifies normal IGES.	yes / no
iges_out_mil_d_28000	Yes - Specifies IGES output that uses the MIL-D-28000 entity subset.	yes / no
iges_out_spl_crvs_as_126	Converts all part geometry spline curves to IGES entity 126 (B-spline) when creating an IGES file.	no / yes
iges_out_spl_srfs_as_128	Converts all part geometry spline surfaces to IGES entity 128 (B-spline) when creating an IGES file.	no / yes
iges_out_start_note	Yes - Exports the text specified by system_iges_header_file and user_iges_header_file as a note on the drawing. To control note placement, use the configuration file option put_iges_drawing_entity.	yes / no
iges_out_symbol_entity	Exports a drawing symbol as an IGES general symbol entity, IGES type number 228 (yes), or as its component entities: notes and lines.	yes / no
iges_out_trim_curve_deviation	Sets the maximum value for the distance between an XYZ trimming curve (boundary) and the underlying surface of a trimmed surface	
iges_out_trim_xyz	Controls whether or not XYZ data is output in addition to UV data for trimmed surfaces.	yes / no
iges_out_trm_srfs_as_143	Yes - Exports all surfaces to IGES entities 141 and 143. This overrides option iges_out_trm_xyz.	no / yes
iges_zero_view_disp	When IGES geometry does not point to any views or the drawing entity. All_views - Creates a copy of the entity for each view. No_views - Does not create the entity at all. As_is - Creates the entity once, using only its own transformation information.	all_views / no_views / as_is
ignore_all_ref_scope_settings	Controls whether object-specific reference scope settings are ignored or not. Environment scope settings will still be enforced. See also allow_ref_scope_change and default_ext_ref_scope	yes / no
ignore_export_geom_declaration	Ignore Export Geometry settings.	yes / no
inch_grid_interval	Modify grid space in inch units for handle movement.	
info_output_format	Setup the default format type for information presentation. Text - information will be output as simple text. HTML - information will be output as html data.	text / html / xml_text / dbg_text
info_output_mode	Sets default method for presenting information. Both - View information on screen and write information to file. Screen - View information on screen only. File - Write information to file only. Choose - Select method from the INFO OUTPUT menu.	choose / screen / file / both

Name	Description	Values
initial_bend_y_factor	Specifies a constant used to determine the neutral bend line for a sheet metal part. This value is always used for non-cylindrical bends. It is used for regular bends only when no bend table is specified.	
instance_search_exhaustive	Specifies extent of instance search. Yes - Include file type when entering instance name in File>Open: type in input panel (e.g. inst_name.prt) or select in drop-down list. If no .idx files are found, only working dir is searched; each generic is not.	no / yes
interface_quality	Sets amount of checking for overlapping lines and collecting lines of same pen color before exporting plot or 2-D file. 0 - No check or collection. 1 - No check, but collection. 2 - Partial check and collection. 3 - Complete check and collection.	
intf_cadds_import_make_solid	Allows the user to solidify all closed quilts during import CADD5 model.	yes / no
intf_cadds_version	Allows the user to switch between the default and alternate converter version for interface with CADD5	10 / 11 / 12 / 13
intf_in_arclength_reparam	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
intf_in_blanked_entities	Filters the import of entities based on their blank status in the import file.	yes / no
intf_in_dwg_pnt_ent	Yes - Converts an IGES or DXF point entity to a drawing point.	yes / no
intf_in_dwg_view	Controls if associativity in an imported IGES view is preserved. 3D_VIEWS - tries to create 3D views if 3D model exists in the file. 2D_VIEWS - Imported IGES views work as 2D. No - Imported IGES views are exploded and become unrelated.	no / 2d_views / 3d_views
intf_in_extract_profiles	Default, no profiles in a data exchange file will be extracted for use.	none / comp / all
intf_in_layer_asm_dialog	Yes - Presents a dialog box to control the import of both layers and assemblies. You can choose from the list of layers and/or solids available in the file to select one or more for import.	yes / no
intf_in_treat_polyline_as	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.	single_polyline / single_spline / set_of_curves / set_of_points
intf_in_use_template_models	Allows the user to use template models in Pro/INTERFACE import.	yes / no
intf_out_as_bezier	Exports all B-splines as Bezier surfaces.	yes / no
intf_out_asm_mapped_item	Export assembly components and instances using MAPPED ITEM entity.	yes / no
intf_out_assign_names	Controls how entity names are handled when an object is exported to STEP format from part or assembly mode. You can give unique names to Pro/ENGINEER datum points, datum axes, datum curves, surfaces, edges, and quilts.	no_name / user_name / id_name
intf_out_auto_layer_ids	Yes - Automatically assigns interface IDs for layers that were not assigned IDs during export. No - Does not assign interface IDs for layers that were not assigned IDs during export.	yes / no
intf_out_blanked_entities	Filters export of entities based on their blank status	yes / no
intf_out_cat_start_model	Specifies Catia start model to be used for CATIA II .model export.	
intf_out_layer	Layer mapping mechanism providing an ability to group entities to layers or blocks.	none / part_layer / block_layer
intf_out_layer_rename_table	Enables you to assign interface ids to layers during export	
intf_out_max_bspl_degree	Controls the maximum degree of exported B-spline surfaces when exporting through IGES	

Name	Description	Values
intf_out_text_length	Sets text length in exported 2-D. As_is - Width of each character is width of strokes. Full_size - Width of each character is width of character text box. Adjusted - Adjusts spacing so no extra space left and end/start character do not overlap.	as_is / full_size / adjusted
intf_use_variable_size	Yes - IGES,DXF,STEP files with variable sized sheet information are imported and put on the appropriate variable-size format. Without a sheet size, the system tries to apply the correct variable-size format. No - The IGES,DXF,STEP drawing is placed on a standard sized format.	no / yes
intf2d_fit_incompatible_data	YES - For import and export, fixes compatibility problems between 2-D external formats (such as IGES and DXF) and Pro/ENGINEER.	yes / no
intf2d_iges_out_hatch	YES - Exports draft xhatching as IGES sectioned area entity. NO - exports draft xhatching as separate geometric entities.	yes / no
intf2d_in_iges_hatch_bnd_layer	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
intf2d_out_cgm_ver	Provides the ability to choose CGM metafile version when exporting from Pro/E drawing.	1, 3
intf2d_out_enhanced_ents	Controls whether the enhanced entities (SPLINE and HATCHING) are converted to the corresponding DXF entities or represented as separate lines and polylines.	spline_and_hatch / spline_only / hatch_only / none
intf2d_out_open_log_window	Yes - Opens Export Log File in special window. No - the Log File will not appear in the window.	yes / no
intf2d_out_pnt_ent	Yes - Exports a drawing point to IGES, DXF or DWG as a point entity. No - Exports a drawing point as shape entities.	yes / no
intf3d_in_close_open_boundaries	Controls fixing of open boundaries on imported surfaces.	yes / no
intf3d_in_enable_layer_join	No - Join surfaces on import as geometry is loaded from file. Yes - Use new method to join surfaces by layer (surfaces contained on the same layer are joined together first) with a better chance of joining those surface elements to form closed geometry.	yes / no
intf3d_in_include_items	Specifies what entity types you want to import from a file.	srfs_crvs_pnts / srfs_crvs / crvs_pnts / srfs_pnts / srfs / crvs / pnts
intf3d_out_cat2_ident_crv	Specifies model parameter IDENT_CRV while exporting to Catia.	
intf3d_out_cat2_ident_pt	Specifies model parameter IDENT_PT while exporting to Catia.	
intf3d_out_cat2_infinity	Specifies model parameter INFINITY while exporting to Catia.	
intf3d_out_cat2_model_sz	Specifies model parameter MODEL_SZ while exporting to Catia.	
intf3d_out_cat2_sag	Specifies model parameter SAG while exporting to Catia.	
intf3d_out_cat2_step	Specifies model parameter STEP while exporting to Catia.	
intf3d_out_datums_by_default	Determines whether datum curves are included when exporting IGES files in Batch mode.	yes / no
intf3d_out_default_option	Controls the type of 3-D data output through Pro/BATCH to an IGES or STEP file.	wireframe / surfaces / wireframe_surfaces / solid / shells / none
intf3d_out_extend_surface	Specifies how surfaces are handled for exporting files to other systems	yes / no
intf3d_out_force_surf_normals	IGES and PDGS export. Controls normals of non-analytic surfaces. Yes - Forces the surface normals to point in a consistent direction. No - Does not force the surface normals to point in a consistent direction.	yes / no

Name	Description	Values
intf3d_out_surface_deviation	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.	
isogen_attribute_map_file	Specifies the mapping between isogen parameter and attribute name.	
isogen_endtype_map_file	Specify name of file mapping Pro/Piping endtypes with ISOGEN endtypes	
isogen_mandatory_attr_file	Specify file containing mandatory attribute filenames used by ISOGEN.	
isogen_nominal_size_map_file	Specify name of file mapping Pro/Piping Size with Nominal Sizes in MM and Inches )	
isogen_output_files_dir	Specify directory to save the PCFs created by ISOGEN interface	
isogen_pcf_filename_format	Specify format to create filenames of PCF files being created	
isogen_symbol_map_file	Specify name of file for mapping Pro/Piping selection names with ISOGEN symbol keys (SKEY)s	
jlink_java2	Option to force use of Java-2 JRE command line ("java" instead of "jre")	off / on
kbd_cmd_abbreviation	On - Enables use of abbreviations when entering menu commands from the keyboard.	on / off
kbd_selection	Provides the ability to select locations in the graphics area via the keyboard. Yes - Selection in the graphics area may be performed via the keyboard. No - Selection in the graphics area requires a mouse.	no / yes
keep_info_datums	Sets the initial state of how the system treats datums and features created at run time.	yes / no
lang_propagate	Specifies language storage options. The option provides flexibility for objects that use a different language than the current	yes / no
let_proe_rename_pdm_objects	Determines whether an object fetched from a PDM database can be renamed in a Pro/Engineer session.	no / yes
linear_tol	An alternate format for setting default linear tolerance dimensions. First value sets the number of decimal places. Second value is the actual tolerance. For example, 6 0.000025 sets tolerance to six decimal places. The default tolerance is 0.000025.	
linear_tol_0.0	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.	
linear_tol_0.00	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.	
linear_tol_0.000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.	
linear_tol_0.0000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.	
linear_tol_0.00000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.	
linear_tol_0.000000	Specify a range for default tolerances of linear and angular dimensions. These values affect only models created after the tolerance options are specified in the configuration file.	
lods_enabled	Determines whether or not the system uses Level of Detail in shaded models during dynamic orientation (panning, zooming and spinning).	no / yes
lods_value	When lods_enabled is Yes, this value specifies the default value used to display LODs. The value is a percentage of the edge size vs the total size of the model.	
maintain_limit_tol_nominal	Maintains the nominal value of a dimension regardless of the changes that you make to the tolerance values.	yes / no

Name	Description	Values
make_parameters_from_fmt_tables	Determines the handling of values entered in a format table when you replace one drawing format with another. Yes - Stores entered values for format table and copies them with the table. No - Prompts you to re-enter all the values for the format table.	yes / no
make_proj_view_notes	Automatically adds view names to projection views in the format "VIEW viewname-viewname." You can modify the view name after adding the view to the drawing.	yes / no
mapkey	Stores a keyboard macro. Use Tools>Mapkeys to create, edit and delete mapkeys.	
mark_approximate_dims	Yes - Displays a tilde (~) in front of a dimension that is set to display as a fraction and has an actual value that is not rounded to equate exactly to the displayed fractional value (the fraction shows 7/32 but the actual value is .22, not 0.21875).	yes / no
mass_property_calculate	Calculate mass properties upon regeneration or by user request.	by_request / automatic
max_animation_time	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.	
maximum_texture_size	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	Specifies the Model Tree configuration file to be loaded when Pro/ENGINEER is started.	
mdo_density	Mass density for dynamics.	
mdo_explicit_integration	Enable explicit integration for dynamic analyses.	yes / no
mdo_spring_wire_radius	Spring wire radius for dynamics spring icons.	
measure_sig_figures	Sets the number of significant figures displayed as results when using the Measure dialog box from the Info menu.	
medusa_2d_config_file	Specifies the Medusa 2D Interface config. file. Absolute or relative path to the file can be used.	
mentor_ver_2_0	Yes - Exports the file in IDF 2.0 format. No - Exports the file in IDF 1.0 format unless IDF 2.0 is selected from the ECAD FORMAT menu.	yes / no
menu_font	Specify font used in the Pro/ENGINEER menu bar, menus, and all their children. Specify the comma-delimited variables in any order (italic bold, 24, times or 24, times, italic bold have the same effect). Any omitted variable uses the standard setting.	
menu_show_instances	Determines whether instance names listed in instance index files appear in file lists.	yes / no
menu_translation	Specifies the language in which the menus display when running non-English version of Pro/ENGINEER.	yes / no / both
merge_smt_srf_without_seam	Yes - no edge left at the merge if the sheetmetal surfaces being merged are same surfaces. No - edge is left even between same sheetmetal surfaces.	yes / no
mesh_spline_surf	Turns the blue mesh surface lines on or off.	no / yes
meter_grid_interval	Modify grid space in meter units for handle movement.	
mfg_auto_ref_prt_as_chk_srf	Yes - In 3, 4, and 5 axis contour and conventional mill sequences, the entire reference part is selected as check surfaces by default when computing the NC sequence toolpath for those sequences. No - You select the check surfaces.	yes / no
mfg_customize_dialog	Yes - Uses the new customize dialog box for customizing NC sequences. No - Uses the Release 19.0 menus.	yes / no
mfg_ijk_num_digits	Sets the number of digits for i, j, k data points in a CL data file.	
mfg_info_location	top left /bottom right -Position of the mfg. Info dialog box.	top_left / bottom_right

Name	Description	Values
mfg_modal_command_file	Enables you to make some of the APTn index topics modal, which means that you repeatedly get a list of major keywords corresponding to this topic upon completing a command line.	
mfg_param_mode	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
mfg_template_dir	Specifies the storage directory for NC templates. Specify full path to avoid problems. NC templates are used in Expert Machinist. Templates store the recipe for a toolpath as the method of tool retrieval.	
mfg_workpiece_transparency	The workpiece transparency is controlled by the value of the configuration option mfg_workpiece_transparency at the time the workpiece is brought into the manufacturing model (assembled or created). You can change its value to a decimal in the range between 0.1 and 1. The bigger the value, the more transparent the workpiece. The default value is 0.5	
mfg_xyz_num_digits	Sets the number of digits for x, y, z data points in a CL data file.	
millimeter_grid_interval	Modify grid space in millimeter units for handle movement.	
min_animation_steps	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.	
minimum_angle_dimension	Controls the minimum angle value under which Pro/ENGINEER assumes colinearity of the selected items when creating a driven dimension.	
model_allow_ref_scope_change	Yes - Users can change the scope setting of components.	yes / no
model_grid_balloon_display	Controls the display of grid balloons.	yes / no
model_grid_neg_prefix	Enables you to specify the prefix to use for negative numbers in the balloon capture of the grid.	
model_grid_num_dig	Controls the number of decimal places displayed in the balloon capture. 0 - Only integers display.	
model_grid_spacing	Sets the default model grid spacing.	
model_note_display	Allows users to display all shown model notes. Applies to shown notes, and does not change Show/Erase status of the notes.	yes / no
model_notes_as_labels	Gives the option to display notes as either full text or as label. Can also be toggled with environment option "Display Notes as Names."	no / yes
model_rename_template	Sets the default renaming convention for copied assemblies. The default new name is the old name appended with an underscore (_). Adding a template such as [*a1*=*b1*] changes every occurrence of a1 to b1 (e.g. TOP_A1_BRACKET copies to TOP_B1_BRACKET).	
model_tree_start	Toggles display of the Model Tree. Yes - Displays the Model Tree window along with its model. No - Does not display the Model Tree window.	no / yes
modelcheck_enabled	Enable ModelCHECK	yes / no
mold_layout_origin_name	Sets a specified coordinate system as the default for the cavity layout origin.	
mold_vol_surf_no_auto_rollback	Do not perform automatic rollback while modifying mold volumes or parting surfaces.	yes / no
motion_envlp_alert	Displays the Motion Envelope alert the first time the quality level is increased.	yes / no
mp_calc_level	Specifies the level of mass properties calculation.	assembly_only / all_models



Name	Description	Values
msg_translation	Specifies language in which Messages are displayed when running non-English version of Pro/ENGINEER.	yes / no
multiple_skeletons_allowed	Allows/restricts multiple skeleton.	yes / no
multipoint_location_count	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.	
native_kbd_macros	Specifies support of keyboard macros written in native language (for example, German).	no / yes
nc_jobman_visible	Yes - make "Submit Path" and "Abort Path" visible. No - make "Submit Path" and "Abort Path" invisible.	yes / no
nccheck_type	Vericut will be used for NC material removal simulation.	vericut / nccheck
ncl_file_extension	Sets the default file extension for Pro/MFG CL data files. Enter the file extension only (without the preceding "."). For example, apt produces filename.apt.	
ncmdl_bar_stock_part_path	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.	
ncmdl_billet_stock_part_path	Sets path for storage and retrieval of stock instances for creating workpiece in NC Model. Specify full path, including part name, to avoid problems.	
ncpost_type	Set to GPOST to access the default post processor supplied with Pro/NCPOST.	gpost / ncpost
new_asm_regen_revnums	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
new_designate_ui		yes / no
new_parameter_ui	Enable the new parameter editor and user interface.	yes / no
new_ref_part_name_disabled		yes / no
new_relation_ui	Enable the new relations editor and user interface.	yes / no
number_user_colors	Specifies the maximum number of different wireframe colors that can display in the graphics area(s) at any given moment.	
number_wireframe_colors	Specifies the maximum number of different wireframe colors that can display in the graphics area(s) at any given moment.	
offset_line_def_color	Specifies the color that offset lines are displayed in drawings.	
offset_line_def_font	Specifies the font in which offset lines are displayed in drawings.	
open_simplified_rep_by_default	Name of rep to retrieve upon file open or "Yes" to always invoke open rep dialog.	
orientation	Establishes the initial standard view orientation. User_default - System uses X-axis and Y-axis values defined by options x_axis and y_axis.	isometric / trimetric / user_default
orthogonal_snap	Controls non-90 degree routing in Pro/DIAGRAM. Yes - Turns the Ortho Snap function on. You can sketch wires only at the default horizontal and vertical orientation. No - Enables you to sketch wires in drawings at angles other than the default.	yes / no
output_mfg_xml	Allows export/import mfg model to/from xml file. Yes - make Export/Import options visible. No - make them invisible.	yes / no

Name	Description	Values
overlays_enabled	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
override_store_back	Forces the storage of objects retrieved from other directories to the current working directory.	yes / no
package_constraints	Action to take for packaged components in Assembly mode. Update - Updates components if assembly changes. Freeze - Does not move components if assembly changes. Disallow - Does not allow packaged components in assembly (must be fully constrained).	disallow / freeze / update
package_ref_alert	Control whether alert dialog is given when packaged component is referenced. Yes - Allow alert when a packaged component is referenced. No - No Alert is given when a packaged component is referenced.	yes / no
param_dec_places	Sets the number of decimal places (0-14) to display in parameters.	
parenthesize_ref_dim	Encloses reference dimensions in parentheses. No - Follows reference dimensions with the text "REF".	yes / no
part_table_editor	Sets the editor that is used to edit part and assembly family tables. Protab - Use Pro/TABLE. Editor - Use the system editor (refer to pro_editor_command).	protab / editor / excel
pdm_rev	Specify how the system parameter, PDM_REV, should be formatted. Only useful for objects submitted to Pro/PDM or Pro/INTRALINK.	pdmrev_fmt_rev_ver / pdmrev_fmt_rev / pdmrev_fmt_branch_rev_ver / pdmrev_fmt_branch_rev
pen_slew	Sets the pen speed for both x and y directions for plotters that are compatible with this option. Consult your plotter's user guide for more information.	
pen_slew_xy	Sets the pen speed for the x and y directions individually for plotters that are compatible with this option. The first value is the x value and the second, the y value. They should be separated by a space.	
pen_table_file	Specifies a default pen mapping table which supersedes other pen mapping information.	
pen1_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen2_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen3_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen4_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen5_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen6_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen7_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	
pen8_line_weight	Sets the weight of entities plotted with pens using an electrostatic plotter. The weight is a range from 1 (thinnest) to 16 (thickest).	

Name	Description	Values
photorender_capability_warnings	Allows the user to disable warnings about PhotoLux only capabilities.	yes / no
photorender_default_height	Sets the default height (in pixels) for the custom size rendered output	
photorender_default_width	Sets the default width (in pixels) for the custom size rendered output	
photorender_memory_usage	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.	
photorender_preview_scale	Allows user defined size ratio at which Pro/PHOTORENDER will render, using the default Render To: Full Window Preview setting. Increasing the value creates higher quality rendering with slower performance. Reducing the value produces reverse results.	
pick_aperture_radius	Specifies the size of the area about the mouse when making selections. Units are 1/1000 of screen size.	
pick_chain_tangent_only	Specifies the extent of draft entity chains selected. Yes - Only entities in the chain that are tangent will be selected. No - All entities connected end to end will be selected.	yes / no
pipe_3d_bend_theor_int_pts	Yes - Displays theoretical intersections and bends in three-dimensional models.	no / yes
pipe_bend_locations_csys_units	Define which assembly units will be used for the piping bend locations calculations. Yes - to use coordinate system assembly units. No - to use pipeline assembly units.	yes / no
pipe_pre_22_bendinfo_conv	Convert bend report information generated prior to Pro/ENGINEER 2000i-2 to current saved analysis format.	yes / no
pipe_solid_centerline	Yes - Display Pro/PIPING centerlines. No - Do not display Pro/PIPING centerlines.	yes / no
pipe_solid_label_format	Specify the pipe solid label format that will be used to automatically generate pipeline labels.	
pipe_update_pre_20_int_pts	Yes - Pipelines created prior to Release 20.0 are updated with intersection point entities. The entities are created in session by Info>Regen Info or when you retrieve the model.	yes / no
pipeline_assembly_library_dir	Sets the default pipeline assembly library directory.	
pipeline_assembly_name_format	Sets the default pipeline assembly name format.	
pipeline_label_format	Specify the pipeline label format that will be used to automatically generate pipeline labels.	
pipeline_solid_start_part_name	Sets the default pipeline solid start part name.	
pipeline_spool_label_format	Specify the pipeline spool label format that will be used to generate the spool labels during pipeline spool generation.	
pipeline_start_assembly_name	Sets the default pipeline start assembly name.	
piping_appearance_map_file	Sets the default piping appearance map filename.	
piping_bolt_nut_select_file	Piping bolts and nuts selection file.	
piping_design_method	Sets the current Piping design method for pipeline modeling.	non_spec_driven / spec_driven / user_driven
piping_end_compatibility_file	Sets the default piping end compatibility filename.	
piping_fitt_angle_tolerance	Sets the default piping fitting angle tolerance value.	
piping_fitt_category_map_file	Sets the default piping fitting category map filename.	
piping_fitt_library_dir	Sets the default piping fitting library directory.	
piping_fluid_parameter	Enables/disables the Parameters tab in Create Pipeline Dialog	no / yes
piping_insulation_dir_file	Sets the default piping insulation directory filename.	
piping_joint_fitting_clearance	Sets the piping joint minimum fitting clearance value.	
piping_joint_fitting_offset	Sets the piping joint minimum fitting offset value.	
piping_manufacture_dir_file	Sets the default piping manufacture directory filename.	

Name	Description	Values
pipng_material_file	Sets the default piping material filename.	
pipng_mcat_dir	Sets the default piping master catalog directory.	
pipng_mcat_dir_file	Sets the default piping master catalog directory filename.	
pipng_min_bolting_clearance	Sets the default piping minimum bolting clearance value.	
pipng_project_data_dir	Sets the default Project data directory.	
pipng_schematic_driven	Set the Schematic Driven Pipeline Modeling ON/OFF	no / yes
pipng_schematic_xml_dir	Specify directory with full path containing schematic information XML file(s).	
pipng_spec_dir_file	Sets the default piping specification directory filename.	
pipng_system_tree_format	Sets the default piping model tree format.	
pipng_thumb_wheel_increment	Sets the default piping thumb wheel increment value.	
pipng_thumb_wheel_ratio_inc	Sets the default piping thumb wheel ratio increment value.	
pipng_wildfire_convert	Always ask before converting old piping assemblies to update the network structure.	manual / never / automatic
plot_file_dir	Specifies the directory where plot files should be written. Use the full pathname to avoid problems. For example /home/users/plotfiles.	
plot_linestyle_scale	Specifies the scaling factor for the DOTFONT line style in a plot.	
plot_names	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
plot_proceed_dialog	Yes - Enables the Plot Proceed dialog which provides a choice for delaying the printing of plot files. This is useful for large plots when using lp print commands. The lp command may not access the plot file before it is removed resulting in no output.	yes / no
plot_to_scale_full_window	Provides the ability to exclude empty window space when plotting the model to scale.	yes / no
plotter	Establishes the default plotter for making plot files.	
plotter_command	Sets the command that you use to start a plot on your system; for Windows NT or Windows 95, windows_print_manager option configures Pro/ENGINEER to plot to a recognized device in the Windows NT or Windows 95 print manager.	windows_print_manager /
plotter_handshake	Specifies the type of plotter handshake initialization sequence generated in the plotter files.	hardware / software
popuphelp_font	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	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
preferred_save_as_type	Allows customizing the order of the Type selections found under File ->Save A Copy.	
prehighlight	Yes - Selectable items beneath the pointer are highlighted prior to being selected. No - Selectable items beneath the pointer are not highlighted prior to being selected.	none / all / geom_only / yes / no
prev_proc_comp_def_color	Specify the default color for components added to a process assembly in a previous step.	
prev_proc_comp_def_font	Specify the default font for previously added components in a process assembly.	
pro_ansys_path	Specifies path to ANSYS executable.	
pro_catalog_dir	Sets the path to the catalog directory containing the catalog menu and the names of other catalog files, such as ejector pins. For example: pro_catalog_dir proe_loadpoint/apps_data/mold_data/catalog. Use full path name to avoid problems.	
pro_cav_lay_rule_dir	Sets the default directory for cavity layout rules. Use the full path name to avoid problems.	

Name	Description	Values
pro_cbltrm_dir	Sets the directory from which terminators are retrieved. Use the full pathname to avoid problems.	
pro_colormap_path	Specifies the directory path for a color map (.map) file to be loaded from disk. Use the full path name to avoid problems.	
pro_crosshatch_dir	Specifies a default directory for your cross-hatch library in which you can save cross-hatching patterns for later retrieval. Its value is the full path name of the default directory.	
pro_dtl_setup_dir	Sets the directory for the drawing setup files. If this option is not set, the system uses the default setup directory.	
pro_editor_command	Enables use of an editor other than the system editor when the option editor has been specified as the value for the options drawing_file_editor and/or family_table_editor and/or relation_file_editor.	
pro_font_dir	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).	
pro_format_dir	Sets the default directory for the drawing format library. Use the full pathname to avoid problems.	
pro_gplug_dir	Specifies the directory in which Granite application plug-ins are located. Use the full pathname to avoid problems.	
pro_group_dir	Sets the default directory for the user-defined feature (UDF) library. If not specified, your current working directory is the default. Use the full pathname to avoid problems.	
pro_insulation_dir	Specifies the directory to search for insulation files used in Pro/PIPEING. Use full path name to avoid problems.	
pro_library_dir	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	Sets the default directory for the part material library. Use the full pathname to avoid problems.	
pro_mdb_dir	Sets the default directory for the Machinability Database (MDB) files. Use the full pathname to avoid problems.	
pro_mf_cl_dir	Sets the default directory for the manufacturing CL data files. Use the full pathname to avoid problems.	
pro_mf_clamp_dir	Specifies the directory where the manufacturing clamps can be found.	
pro_mf_param_dir	Sets the default directory for the manufacturing parameter files. Use the full pathname to avoid problems.	
pro_mf_tprm_dir	Sets the default directory for the manufacturing tool files. Use the full pathname to avoid problems.	
pro_mf_workcell_dir	Sets the default directory for the manufacturing workcell files. Use the full pathname to avoid problems.	
pro_mfg_cmdsyn_dir	Sets the default directory for the syntax menu files in Pro/MFG. Use the full pathname to avoid problems.	
pro_nastran_path	Specifies the path to the MSC/NASTRAN executable (nastran).	
pro_note_dir	Specifies the directory from which to retrieve notes entered from a file. Use the full path name to avoid problems.	
pro_palette_dir	Sets the default directory for drawing symbol palette files.	

Name	Description	Values
pro_pip_fitt_dir	Specifies the directory to search for fittings used in Pro/PIPING.	
pro_pip_instk_dir	Specifies the directory to search for linestock files used in Pro/PIPING. Use full path name to avoid problems.	
pro_plot_config_dir	Sets the directory of your user-defined plotter configuration file. Use the full pathname to avoid problems.	
pro_sheet_met_dir	Sets the default directory of your user-defined bend tables. If not set, the Pro/SHEETMETAL supplied bend tables are used. Use the full path name to avoid problems.	
pro_smt_params_dir	Specifies the directory to save/retrieve Sheet metal parameters files. Use full path name to avoid problems.	
pro_solver_name	Specifies the name of the user-defined solver to be included in the SOLVE menu. See also <u>pro_solver_path</u> .	
pro_solver_path	Specifies the path to the user-defined solver whose name is defined in the SOLVE menu. See also <u>pro_solver_name</u> .	
pro_spool_dir	Sets the directory from which spools are retrieved. Use the full pathname to avoid problems.	
pro_stheno_command	Specifies the command for Stheno launch.	
pro_surface_finish_dir	Sets the default directory for your user-defined surface finish symbols. Use the full pathname to avoid problems.	
pro_symbol_dir	Set and automatically create the default directory for saving and retrieving user-defined symbols. If you do not specify a directory, the system uses the current working directory. Use full path name to avoid problems.	
pro_texture_library	Allows the user to specify a different graphics library.	
pro_unit_length	Sets the default units for new objects.	unit_inch / unit_foot / unit_mm / unit_cm / unit_m
pro_unit_mass	Sets the default units for mass for new objects.	unit_ounce / unit_pound / unit_ton / unit_gram / unit_kilogram / unit_tonne
pro_unit_sys	Specifies the default system of units for new models.	mks / cgs / mmns / fps / ips / proe_def
pro_weld_params_dir	Specifies the directory to search when a weld parameter file is needed.	
prodevdat	Specify the name of the Pro/Develop registry file.	
profile_output_utils	Affects CL-Data output for profile milling sequences. Yes - Sequence parameter output_point is available with options: tip - Toolpath is output with respect to the tool center; on_profile - Toolpath is output with respect to the profile of the tool.	yes / no
prompt_on_erase_not_disp	Specifies whether or not to display a prompt allowing you to save your undisplayed objects. This option is used in conjunction with the Erase > Not Displayed command on the File menu.	yes / no
prompt_on_exit	Determines if you are prompted to save objects when exiting a Pro/ENGINEER session.	yes / no
proncpst_dbf	Enter directory path to NCPost database file.	
propagate_change_to_parents	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



Name	Description	Values
prorembatch_queue_manager	Enables toolpath computation on a separate host machine. Required values are Processor name, Remote machine's hostname, Type of transport (always 0), RPC processor number (same as that of remote processor), Version of queue manager.	
protable_in_background	Yes - Starts Pro/TABLE editor in the background. Reduces time between pressing a menu button that uses Pro/TABLE and entering the data.	no / yes
protkdatt	Specify the Pro/TOOLKIT registry file.	
provide_pick_message_always	No - A message describing the (pre)selected item is provided only while querying. Yes - A message describing the selected item is provided in all cases (i.e. even when Query is not used).	yes / no
ps_output_with_pagesize	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
ptncpost_dbf	Enter directory path to NCPost database file.	
punch_axis_points	Yes - Enables the creation of punch axis points in Sheetmetal cuts and punches.	yes / no
put_iges_drawing_entity	Specifies whether or not to suppress the output of the IGES drawing entity, #404, to the IGES file. No - The drawing entity is not output.	yes / no
radial_hole_linear_dim	YES - Option for creation of radial hole with a linear dimension will be available.	yes / no
raster_plot_dpi	Determines the resolution for plotter outputs variables are in dots per inch.	100 / 200 / 300 / 400
ratio_grid_interval	Modify grid space in percentage units for handle movement.	
read_famtab_file_on_retrieve	No - Ignores filename.ptd. Yes - Creates and saves filename.ptd and uses that file when retrieving the generic.	no / yes
read_vda_in_pset_as_spline	Specifies whether a VDA Pset entity is imported to Pro/ENGINEER as a spline.	yes / no
recompute_iges_dim_value	Indicates whether to recompute associative dimensions upon import of an IGES file.	yes / no
ref_scope_copy_color	Set color of backup references.	
ref_scope_copy_color_change	Enable changing colors of backup references during selection.	no / yes
ref_scope_no_pick_to_copy	Disable selecting references backup by references control.	no / yes
ref_scope_no_pick_to_prohibit	Disable selecting references prohibited by references control.	no / yes
ref_scope_prohibit_color	Set color of prohibited references.	
ref_scope_prohibit_color_change	Enable changing colors of prohibited references during selection.	no / yes
regen_backup_directory	Specifies the directory in which the system stores regen_backup models. (See regen_backup_using_disk.) The primary default is the current directory, but if that directory is read-only, the secondary default is the /tmp directory.	
regen_backup_using_disk	Specifies whether or not the system backs up the current model to disk before each regeneration.	yes / no
regen_int_mdls_on_retr	Yes to regen models intersected with assem feats during assembly retrieval.	yes / no
regen_layout_w_assem	Specifies whether or not the layout will be automatically regenerated upon assembly regeneration.	yes / no
regenerate_read_only_objects	Determines whether an object fetched from a Pro/INTRALINK database with blocking status can be regenerated in a Pro/Engineer session.	yes / no
relation_file_editor	Sets the editor used to edit relations.	protab / editor
remember_last_get_point_option	Yes - Sets the default of the GET POINT menu as the last option picked among Pick Pnt, Vertex and On Entity.	yes / no

Name	Description	Values
rename_drawings_with_object	Controls whether the system copies associated drawings automatically with parts and assemblies.	none / part / assem / both
replace_comp_name_method	Sets the naming convention used when replacing components.	transfer / remove / preserve
replace_comp_notes_method	Specify treatment of notes when components are replaced. Transfer - transfer to new component. Remove - remove the notes. Preserve - save notes in case component is restored.	transfer / remove / preserve
reserved_menu_space	Reserves space for extra menus to the right of the Pro/ENGINEER window (1.0-4.0). This option overrides windows_scale.	
restricted_gtol_dialog	Determines whether the Geometric Tolerance dialog restricts the user by graying out elements that are considered "illegal".	yes / no
retain_display_memory	Determines if the display of an object currently on the screen is kept in memory when you quit the window. Doing so (yes) makes the retrieval of objects in memory faster.	yes / no
retrieve_data_sharing_ref_parts	Automatic reference parts retrieval for dependent data sharing features.	yes / no
right_mouse_button_popup	Specify whether or not the right mouse button pop up menu is available or not.	yes / no
rotate_postscript_print	Specifies rotation for printing. Yes - Rotates a PostScript plot 90 degrees counterclockwise. Use when you are plotting a landscape drawing on a portrait-type laser printer, or a portrait drawing on a landscape-type laser printer.	yes / no
save_bitmap	When performing a save on a model, determines what level of the assembly will have bitmap images created of modified components.	none / toplevelonly / alllevels
save_bitmap_type	When performing a save on a model, determines the type of image file that is created for that model. Used with "save_bitmap" option.	bmp / tiff / gif / cgm / jpeg
save_display	Yes - Stores view geometry and detail items such as solid dimensions. These items are displayed when retrieving the drawing in View-Only mode.	no / yes
save_drawing_picture_file	Embed - Embeds a picture file inside a drawing for preview purposes. Export - Saves a drawing file as a picture file in the working directory when saving a drawing. Both - Does both embed and export.	no / export / embed / both
save_instance_accelerator	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
save_model_display	Sets amount of graphical data stored. Wireframe - Wireframe data. Shading_high - Most detail (shaded). Shading_med - Medium detail (shaded). Shading_low - Least detail (shaded). Shading_lod - Detail determined by View Performance dialog (shaded).	no_display / wireframe / shading_low / shading_high / shading_lod
save_modified_draw_models_only	Determines whether the system saves the model after you have changed it. No - Saves the model every time that you store the drawing.	no / yes
save_object_in_current	Use with the override_store_back option. Yes - Objects retrieved from a directory where you do not have write permission will be stored in the current directory. No - These objects are not saved at all.	yes / no
save_objects	Determines when an object and its dependent objects (such as a part used in an assembly) are stored.	all / changed / changed_and_specified / changed_and_updated
save_triangles_flag	Spinning shaded images with software graphics only. Note: This option has no effect if your system uses hardware graphics.	no / yes

Name	Description	Values
save_unchanged_pdm_object	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
scope_invalid_refs	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
search_path	Specifies list of directories to search (in order) for object/file retrieval. These directories, the working directory, and directories in search.pro file (refer to search_path_file) are Pro/ENGINEER's search path. Use full path name to avoid problems.	
search_path_file	Specifies path to text file search.pro (a list of directory path names). The directories listed in search.pro, the working directory, and directories specified by search_path are Pro/ENGINEER's search path. Use full path name to avoid problems.	
section_color	Specifies the color of sketched sections.	default / drawing_color
sel_insts_on_comp_retrieval	Yes - If the components used in the assembly instances are themselves generics, the systems asks if you want to choose an instance as each component is retrieved. No - The generic model of the component is retrieved automatically.	yes / no
select_hidden_edges_in_dwg	Controls selectability of hidden edges in no-hidden drawing views.	yes / no
select_on_dtm_edges	All_modes - Select a datum plane by clicking on its visual boundary. Sketcher_only - Select a datum plane by clicking on its tag in modes other than Sketcher. If you use Query frequently, set this option to Sketcher_only.	all_modes / sketcher_only
selection_of_removed_entities	Yes - Entities in front of cross-section (planar or offset) can be selected, clipped (using Z-clipping), or erased with EDGE DISP menu. No - Geometry, datum points or curves, cosmetic features, threads, grooves, or coordinate systems cannot be erased.	yes / no
set_menu_width	Determines width of menu manager panels. Default value is equivalent to 8 character widths. Acceptable range is from 8 to 20. Values lower than 8 and higher than 20 are ignored.	
set_model_types_to_save	Determines which model types are used in the Save Changed command. The format is a list of extensions separated by spaces.	
set_trail_single_step	Yes - Enables a trail file to be single-stepped by entering .	yes / no
shade_mfg_ref_models	No - Does not process reference parts when generating the shaded image (to save time). Yes - Shades reference parts when generating shaded images.	no / yes
shade_moving_comp	Yes - Shades assembly components while they are being moved. No - Displays components in wireframe while they are being moved.	yes / no
shade_surface_feat	Controls the shading of surface features.	yes / no
shade_windows	Enables shading of objects in all windows. This option does not also enable color shading.	all_windows / one_window
shade_with	Controls the display of datum curves on shaded objects.	curves / no
shaded_postscript_background	Specifies how background colors are handled in Postscript files of shaded objects.	yes / no
show_all_mold_layout_buttons	Sets all mold layout buttons visible.	yes / no
show_axes_for_extr_arcs	Determines if axes are created for newly extruded arcs.	yes / no
show_cav_fit_loc_sys	Allows the printing of the transformation matrix of a Pro/Verify Fit into a file.	yes / no
show_dim_sign	No - Dimension values always appear positive. Negative value creates geometry to opposite side. Yes - Entering negative value for dim displayed as negative creates geometry to same side. Dims for csys and datum point offsets always show neg/pos value.	yes / no

Name	Description	Values
show_geom_checks_on_creation	Yes - The SHOW ERRORS menu appears at the end of feature creation when the feature being created has Geometry Checks.	yes / no
show_preview_default	This option determines the default behavior for preview in Show/Erase.	keep / remove
show_shaded_edges	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
shrinkage_value_display	Determines how dimensions are displayed when shrinkage is applied to a model with Pro/MOLDESIGN. Percent_shrink - Dimensions appear with the shrinkage percent.	final_value / percent_shrink
shrinkwrap_alert	Displays the Shrinkwrap alert the first time the quality level is increased.	yes / no
silhouettes	Specifies whether silhouette edges are displayed or not by default.	no / yes
sim_asm_modeling	Controls automatic creation of assembly links and free connections in Pro/MECHANICA. The default is YES.	yes / no
sim_display_arrow_scale	Toggles automatic arrow scaling.	yes / no
sim_display_arrow_tail_touching	Toggles display of Load arrows to have tails or heads touching.	yes / no
sim_display_beam_releases	Toggles display of Beam Release icons.	yes / no
sim_display_beam_sections	Toggles display of beam section icons.	yes / no
sim_display_beams	Toggles display of beam icons.	no / yes
sim_display_condensation_intfs	Toggles display of condensation interfaces icons.	yes / no
sim_display_contact_regions	Toggles display of Contact Region icons.	yes / no
sim_display_gaps	Toggles display of gap icons.	no / yes
sim_display_in_spin	Toggles display of simulation objects in Dynamic Spin.	yes / no
sim_display_interfaces	Toggles display of structure connection icons.	no / yes
sim_display_load_colors	Toggles display of Load colors.	yes / no
sim_display_load_distribution	Toggles display of distributed load vectors over entire entity.	yes / no
sim_display_load_icons	Toggles display of load icons.	yes / no
sim_display_load_value	Toggles display of Load values.	yes / no
sim_display_masses	Toggles display of mass icons.	no / yes
sim_display_matl_assignments	Toggles display of Material Assignment icons.	yes / no
sim_display_mesh_and_model	Toggles display of Mesh together with Model.	yes / no
sim_display_mesh_entities	Toggles display of Mesh entities.	yes / no
sim_display_mesh_mode	Specifies the mode of mesh display in FEM preprocessor.	no_mesh / wireframe / hidden / nohidden / shading
sim_display_mesh_shrink_elems	Controls display of Mesh through a specified percentage of shrinking of its elements.	
sim_display_motion_cams	Toggles display of motion cam icons.	yes / no
sim_display_motion_connections	Toggles display of motion connection icons.	yes / no
sim_display_motion_contact_rgns	Toggles display of motion contact region icons.	yes / no
sim_display_motion_design_vars	Toggles display of design variable icons.	yes / no
sim_display_motion_drivers	Toggles display of motion driver icons.	yes / no
sim_display_motion_gears	Toggles display of motion gear icons.	yes / no
sim_display_motion_ground_pnts	Toggles display of motion ground point icons.	yes / no
sim_display_motion_lcs	Toggles display of local coordinate systems icons.	yes / no
sim_display_motion_loads	Toggles display of motion load icons.	yes / no

Name	Description	Values
sim_display_motion_slots	Toggles display of motion slot icons.	yes / no
sim_display_names	Toggles display of Simulation Object names.	yes / no
sim_display_rigid_connections	Toggles display of rigid connection icons.	yes / no
sim_display_rigid_dof	Toggles display of Rigid Link DOF icons.	yes / no
sim_display_rigid_links	Toggles display of Rigid Link icons.	yes / no
sim_display_shells	Toggles display of shell icons.	no / yes
sim_display_spot_welds	Toggles display of spot weld icons.	yes / no
sim_display_springs	Toggles display of spring icons.	no / yes
sim_display_struct_constraints	Toggles display of structure constraint icons.	yes / no
sim_display_struct_loads	Toggles display of structure load icons.	yes / no
sim_display_therm_bcs	Toggles display of Thermal Boundary Condition icons.	yes / no
sim_display_therm_loads	Toggles display of thermal load icons.	yes / no
sim_display_weighted_dof	Toggles display of Weighted Link DOF icons.	yes / no
sim_display_weighted_links	Toggles display of Weighted Link icons.	yes / no
sim_display_welds	Toggles display of Weld icons.	yes / no
sim_fatigue_biaxiality_correct	Controls the use of biaxiality correction. The default value is YES.	yes / no
sim_fatigue_biaxiality_method	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
sim_fatigue_confidence_level	Specifies the percentage confidence in the predicted life result value. The default value is 90, and range is from 0.1 to 99.9	
sim_fatigue_external_matdata	Controls the use of external material data. The default value is NO.	no / yes
sim_fatigue_hysteresis_gate	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.	
sim_fatigue_infinite_life_value	Specifies the value for the lives beyond cutoff. The default value is 1e20 and range is from 1e15 to 1e30.	
sim_fatigue_mean_stress	Controls the application of the mean stress correction. The default value is YES.	yes / no
sim_fatigue_mean_stress_method	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
sim_fatigue_safety_margin	Specifies a factor used to determine the Life Confidence quantity. The default value is 3 and range is from 1.1 to 100.	
sim_fatigue_user_directory	Specifies the directory for user files (for example, external material files). The default value is the current directory.	
sim_fatigue_write_surf_stress	Controls writing of surface stresses to a neutral file. The default value is NO.	no / yes
sim_fem_nastran_use_pshell	Yes - use PSHELL cards, No - use PCOMP cards for laminate composite shells in NASTRAN output file.	yes / no
sim_load_mech_mesh	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
sim_mat_poissons_notation	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

Name	Description	Values
sim_max_memory_usage	Specifies the amount of memory in megabytes to be used in Mechanica run and FEM mesher. The default value is 128.	
sim_motion_analysis_accuracy	Specifies the default accuracy to which Pro/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.	
sim_motion_analysis_assem_tol	Specifies the default error allowed for the assembly analysis. The default value is 0.001.	
sim_motion_analysis_duration	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.	
sim_motion_analysis_increment	Specifies the default increment between steps at which Pro/MECHANICA reports the motion of the mechanism. The default value is 0.1	
sim_motion_analysis_integrator	Controls the default integrator Pro/MECHANICA uses when running the motion analysis. The value types are IMPLICIT (default) and EXPLICIT.	implicit / explicit
sim_motion_analysis_method	Controls the default method Pro/MECHANICA uses to analyze the motion of your model. The default is FASTEST_AVAILABLE.	fastest_available / time_integration / kinetostatic
sim_motion_analysis_start_time	Specifies the default time in seconds at which the motion analysis begins. The default value is 0.	
sim_motion_analysis_vel_tol	Specifies the default error allowed for the velocity analysis. The default value is 0.001.	
sim_motion_output_fly_file	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
sim_nastran_use_coupmass	Add PARAM,COUPMASS,1 in the Nastran deck file to enforce consistent mass matrix generation.	yes / no
sim_output_ids_for_layers	Outputs FEM mesh element IDs on each layer into XML file.	yes / no
sim_output_obj_names	Output Beam Sections names, Analyses names, Coordinate Systems names as comments.	yes / no
sim_pp_background_color	Specifies the background color of the work area for the results display. The default value is PROE.	proe / blue / black / white
sim_pp_vrml_export_format	Flags to denote VRML1.0 or VRML2.0 (default) output.	vrml2.0 / vrml1.0
sim_pp_vrml_feature_edges	Flag to enable the inclusion of feature edges in the VRML file. The default value is NO.	no / yes
sim_regen_on_entry	Yes - Regenerate model on entrance to Simulation Application. No - Don't regenerate the model.	yes / no
sim_run_copy_fem_neutral_file	Controls whether the FEM neutral file (with extension .fnf) is copied into the study directory. The default value is YES.	yes / no
sim_run_out_dir	Specifies the directory for Pro/MECHANICA run temporary storage. By default, the current directory is used.	
sim_run_tmp_dir	Specifies the directory for Pro/MECHANICA run output storage. By default, the current directory is used.	
sim_use_layers	Allows layers to become groups for Pro/MECHANICA Independent mode.	no / yes
simulation_fem_mode	Yes - Activates FEM mode. No - Allows access to Mechanica only.	yes / no
skeleton_model_default_color	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.	



Name	Description	Values
sketcher_animated_modify	No - The section is not animated as modifications are regenerated.	yes / no
sketcher_blended_background	This option controls whether the blended background should be used in 3D Sketcher. The option has no effect if blended background is turned off generally.	no / yes
sketcher_collinear_skamps	Yes - collinear constraints will be used by Intent Manager; No - collinear constraints will not be used by Intent Manager.	no / yes
sketcher_dec_places	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.	
sketcher_disp_constraints	Shows the constraints when sketching in Sketcher Mode; for example, H for horizontal, V for vertical, and so on.	yes / no
sketcher_disp_dimensions	No - Suppresses the display of all dimensions while in sketcher.	yes / no
sketcher_disp_grid	Specify whether or not to display the sketcher grid.	yes / no
sketcher_disp_vertices	No - Suppresses the display of yellow points on all vertices while in sketcher.	yes / no
sketcher_disp_weak_dimensions	Yes - weak dimensions will be displayed; No - weak dimensions will not be displayed.	yes / no
sketcher_equal_length_skamps	Yes - equal length constraints will be used by Intent Manager; No - equal length constraints will not be used by Intent Manager.	no / yes
sketcher_equal_radii_skamps	Yes - equal radii constraints will be used by Intent Manager; No - equal radii constraints will not be used by Intent Manager.	no / yes
sketcher_grid_angle	Enter a grid angle value to override the default grid angle value.	
sketcher_intent_manager	Specifies which sketcher to use. Yes - Intent Manager is the default. No - Old sketcher is the default.	yes / no
sketcher_lineup_hor_skamps	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
sketcher_lineup_ver_skamps	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
sketcher_lock_modified_dims	Yes - modified dimensions will be locked; No - modified dimensions will not be locked.	yes / no
sketcher_midpoint_skamps	Yes - midpoint constraints will be used by Intent Manager; No - midpoint constraints will not be used by Intent Manager. )	no / yes
sketcher_num_digits	Enter a number of digits for scetcher accuracy.	
sketcher_parallel_skamps	Yes - parallel constraints will be used by Intent Manager; No - parallel constraints will not be used by Intent Manager.	no / yes
sketcher_perpendicular_skamps	Yes - perpendicular constraints will be used by Intent Manager; No - perpendicular constraints will not be used by Intent Manager.	no / yes
sketcher_point_on_entry_skamps	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
sketcher_refit_after_dim_modify	Refits section after dimension modification in 2D section or when creating the first feature.	yes / no
sketcher_rel_accuracy	Enter a sketcher relative accuracy.	
sketcher_same_point_skamps	Yes - same points constraints will be used by Intent Manager; No - same points constraints will not be used by Intent Manager.	no / yes
sketcher_save_preview_image	This option controls whether section files should be saved with embedded image information which can be used to preview sections in File/Open dialog.	no / yes

Name	Description	Values
sketcher_set_grid_method	"Automatic" - grid spacing will be automatically set, "Manual" - grid spacing will be manually set.	automatic / manual
sketcher_set_grid_x_spacing	Enter a x-grid spacing value to override the default x-grid spacing value.	
sketcher_set_grid_y_spacing	Enter a y-grid spacing value to override the default y-grid spacing value.	
sketcher_starts_in_2d	Defines initial model orientation in Sketcher mode. Yes - (2D orientation) Looking directly at section (sketching) plane. No - (Orientation unchanged) Sketch directly on the 3D part.	yes / no
sketcher_symmetric_skamps	Yes - symmetric constraints will be used by Intent Manager; No - symmetric constraints will not be used by Intent Manager.	no / yes
sketcher_tangent_skamps	Yes - tangent constraints will be used by Intent Manager; No - tangent constraints will not be used by Intent Manager.	no / yes
sketcher_undo_stack_limit	Sketcher saves a copy of each function performed. The number of possible saved functions depend on the number specified in the option. The undo menu can be used to remove the stored functions.	
smooth_lines	Choose whether to smooth lines by default or not.	no / yes
smt_bend_notes_dflt_display	Define the default state of Bend Note display (sheetmetal Mode)	yes / no
smt_bend_notes_direction_down	Define the Down Direction symbol for Bend Notes (sheetmetal Mode)	
smt_bend_notes_direction_up	Define the Up Direction symbol for Bend Notes (sheetmetal Mode)	
smt_bend_notes_order	Define the Fields Order for Bend Notes (sheetmetal Mode)	
smt_bend_notes_type_formed	Define the Formed symbol for Bend Notes (sheetmetal Mode)	
smt_bend_notes_type_rolled	Define the Rolled symbol for Bend Notes (sheetmetal Mode)	
smt_crn_rel_display	Yes - Corner relief notes will be displayed. No - Corner relief notes will not be displayed.	no / yes
smt_mp_method	MASS - Suppressed Flat Pattern and Flatten Forms will be temporary resumed before mass properties calculation of sheetmetal part. CG - Mass Properties calculation will be performed on current state of sheetmetal part. BOTH - Both methods will be used one after the other.	mass / cg / both
smt_outside_mold_lines	Yes - Outside Mold Lines will be created during Flat Pattern creation.	yes / no
spin_center_display	Determines whether or not the spin center symbol displays.	yes / no
spin_control	Specifies spin mode of an object. Click - Pick a place on the scale to reorient model to this position. Drag - Pick on current value and drag it along scale to change model orientation continuously. To stabilize orientation angle, pick on scale again.	drag / click
spin_rate_zoom_sensitive	Allows slower spinning when zoomed in on a model.	yes / no
spin_with_notes	Yes - 3D feature and model notes are displayed during dynamic spinning of a model.	yes / no
spin_with_part_entities	Yes - Datum features will be displayed during dynamic spinning of a model.	yes / no
spin_with_silhouettes	Yes - Silhouette lines will be displayed during dynamic spinning of a model.	no / yes
start_appmgr	Determines whether or not the application manager starts when Pro/ENGINEER is started.	yes / no
start_model_dir	Provides the complete path to the directory containing start parts and assemblies.	
std_nastran_template	Sets the file path of a NASTRAN deck template. Use full path to avoid problems.	
step_appearance_layers_groups	No - Output of Appearances, Layers, and Groups is enabled for STEP standard AP214. Yes - Output of Appearances, Layers, and Groups is enabled for STEP standard AP203.	yes / no
step_export_ap214_asm_def_mode	Toggles the default setting of the File Structure field in the STEP Export dialog box for supported assembly STEP export files.	single_file / separate_parts_only / separate_all_objects

Name	Description	Values
step_export_dwg_views	AS_3D_VIEWS - Export 3-D model geometry with its associative views. AS_3D_VIEWS_ASSOC_DRAFT - Export 3-D model geometry with its associative views and view related annotations. NO - Export only 2-D representations of the 3-D models. This is the default.	no / as_3d_views / as_3d_views_assoc_draft
step_export_format	Specifies output format when exporting to STEP. AP203_IS - Formats output to ISO 10303 AP203IS format. AP214_CD - Formats output with geometry that meets the specification for the long-term schema for AP214cc1.	ap203_is / ap203_dis / ap214_cd / ap202_is / 203_is_ext / ap214_dis / ap209_dis
store_tool_path_file	Specifies how tool path information is computed. Yes - Tool path information stored in file filename.tph (filename is manufacturing model name). No - Tool path is computed each time manufacturing model is retrieved or parameters are changed.	yes / no
style_default_connections	Yes - Add connections to a newly created Style surface.	yes / no
summary_help	Refers to the one-line help Messages that are either displayed or not displayed.	yes / no
suppress_dlg_license_expired	Suppresses "License expired: List of options and expiration days" dialog	
suppress_license_loss_dialog	Yes - Suppresses the 'Regained Floating License' dialog box. No - The 'Regained Floating License' dialog box will display.	yes / no
switch_dims_for_notes	Yes - Dimensions will be displayed in their symbolic format during drawing note creation. No - Dimensions will be displayed in drawing notes as numerical values.	yes / no
sym_leader_orient_move_text	With Pro/DETAIL, automatically regroupes weld symbol instances after you move the text.	yes / no
symbol_instance_palette_file	Specifies the location of the symbol instance palette.	
symbol_palette_input	Controls the display of the special symbol palette during note creation.	yes / no
system_background_color	Defines default graphics area background color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_colors_file	Specifies the system color file. Use the full path name to avoid problems.	
system_curves_color	Defines default curve color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_dimmed_menu_color	Defines default secondary highlight color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_edge_high_color	Defines default edge highlight color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_geometry_color	Defines default color of solid wireframe entities. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_hidden_color	Defines default color of hidden, wireframe entities. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	

Name	Description	Values
system_highlight_color	Defines default primary highlight color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_iges_header_file	Inserts specified text file into the Start section of IGES files (refer to user_iges_header_file). When used with user_iges_header_file, the system text appears first.	
system_letter_color	Defines default color of datum tags. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_section_color	Defines default section color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_sheetmetal_color	Defines default color of Sheetmetal parts. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
system_volume_color	Defines default color of Manufacturing volumes. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using View>Display Settings>System Colors.	
tablet_device_name	Specifies the name of a digitizing tablet.	
tan_angle_for_disp	For display purposes only. Sets the angle between two quilt surfaces below which the edge will be displayed as tangent. Enter an angle between 1.5 and 15 degrees.	
tangent_edge_display	Determines how edges between tangent surfaces are displayed.	no / solid / centerline / phantom / dimmed
template_designasm	Specifies the designated template assembly. Use full path to avoid problems.	
template_drawing	Specify the drawing used as the default drawing template.	
template_ecadasm	Specify the model used as the default ECAD assembly template.	
template_ecadpart	Specify the model used as the default ECAD part template.	
template_mfgcast	Specify the model used as the default manufacturing cast template.	
template_mfgmold	Specify the model used as the default manufacturing mold template.	
template_mold_layout	Specify the mold layout assembly used as a default template.	
template_sheetmetalpart	Specify the model used as the default sheetmetal part template.	
template_solidpart	Specify the model used as the default part template.	
terminal_command	Specifies the full path to the terminal emulator command (the command that starts shell windows). Use the command for your system. Enter the complete path name and the terminal command.	
texture	Yes - Applied textures will be displayed when the model is shaded.	no / yes
texture_search_path	Appends a directory to the search path for textures.	
thermo_position_hint	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
tiff_compression	Determines whether or not to-tiff exports are compressed using G4 compression	none / g4
tiff_type	Determines the type of tiff items are exported to variables refer to color sets	rgb / palette / grayscale / mono

Name	Description	Values
todays_date_note_format	Controls the initial format of the date displayed in a drawing. The format for the setting is a string consisting of three portions: the year, the month, and the date. You can enter the portions in any order.	
tol_display	Displays dimensions with or without tolerances.	yes / no
tol_mode	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
tolerance_class	Sets the default tolerance class for ISO-standard models. The system uses the tolerance class in conjunction with the dimension value when retrieving tolerances for General or Broken Edge dimensions.	fine / medium / coarse / very_coarse
tolerance_standard	Sets the tolerance standard used when creating the model.	ansi / iso
tolerance_table_dir	Sets the default directory for user-defined tolerance tables for ISO-standard models. All Holes and Shafts tables overwrite existing tables when loaded.	
toolkit_registry_file	Tells Pro/E the full path to which toolkit registry file to be used. This option replaces the R17 option prodevdat.	
topobus_enable	Enables usage of the Associative Topology Bus for exchanging data.	yes / no
tracesketch		no / yes
trail_delay	Sets a delay in seconds between trail file steps.	
trail_dir	Instructs Pro/ENGINEER to create its trail file in the specified directory rather than in the startup directory.	
train_file_path	Specifies the default directory for training files. The path name must start from the root of the directory, for example, /home/users/library/training.	
transparency	Yes - Transparent colors will appear as such when the model is shaded.	no / yes
units_system_dic_file	Specifies the file containing the mapping for MCAT filename and units.	
update_harness_mfg_line_color	Update cable line color for harness-mfg part created in release 2000i or earlier.	yes / no
update_pre_130_harness	Yes - When a cable is added between two previously laid out locations in a three-dimensional reference harness, display of the new cable will update automatically in pre-Release 13 flattened cable models.	yes / no
update_pre_16_cable_layers	Yes - Harnesses containing cables routed prior to Release 16.0 will be automatically updated upon retrieval to comply with new display standards.	yes / no
update_rep_refs	"Yes" updates top assembly simplified reps for replacements during regeneration and upon replacement.	yes / no
use_8_plotter_pens	Specifies whether to support up to 8 plotter pens. The initial default is 4 pens.	no / yes
use_cadam_plot_data	Determines whether the information in the Plot Axis System element should be taken into account when importing a CADAM drawing.	yes / no
use_export_2d_dialog	Yes - Opens export options dialog when exporting Pro/E drawing. No - exports file without the options dialog.	yes / no
use_iges_font_1003	Used to disable the use of IGES font 1003.	yes / no
use_iges_kanji_font_2001	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

Name	Description	Values
use_major_units	Determines if fractional dimensions are displayed in terms of feet-inches or meter-mm. Yes - The major units are used. For example, when the units are inches and 25.125 is converted to a fraction, the dimension becomes 2' 1-1/8".	yes / no
use_master_for_bad_disp_reps	Yes to retrieve master rep when graphics rep has no display data.	yes / no
use_master_for_bad_disp_reps	Yes to retrieve master rep when graphics rep has no display data.	yes / no
use_new_intchg	"yes"-Pro/ENGINEER uses the consolidated interchange assembly functionality. "no"-Pro/ENGINEER uses the separate functional interchange assemblies and simplify interchange assemblies.	yes / no
use_nom_dim_val_in_expr	Yes - use dimensions' nominal values in expressions. No - use current values.	yes / no
use_part_color_for_hidden_lines	Determines how to color the hidden lines of parts that have user defined colors. "YES" - Use the dimmed part color for hidden lines. SYSTEM_HIDDEN_COLOR is ignored if it has been specified.	no / yes
use_pre_wildfire_text_font	Yes - Use the old stroke-based PTC font as the default font. No - Use a True Type font as the default font.	no / yes
use_software_linefonts	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
use_temp_dir_for_inst	Explicitly makes Pro/ENGINEER use the Temp directory for regenerating instances of models.	no / yes
user_defined_grid_interval	Modify grid space in user defined length units for handle movement.	
user_iges_header_file	Filename - Insert specified text file into the Start section of IGES file. Valid parametric note symbols will be replaced during export. For example, \&dwg_name in the text file is replaced by the drawing name when a drawing is exported.	
variable_plots_in_inches	No - Variable plot sizes may be entered in millimeters.	yes / no
variant_drawing_item_sizes	No - Drawing items moved/copied to a different sheet or located on a changed sheet keep same size and relative orientation on paper. Yes - Some items scale/reposition to be the same on paper, and others scale and/or reposition to be the same on screen.	yes / no
vda_header	The complete filename of the text file containing VDA header information. If you intend to use the same header for all VDA files, specify the full pathname.	
versatec_cutter_installed	Yes - Indicates that a cutter is installed on a Versatec plotter.	no / yes
visible_mapkeys	No - Mapkeys will run without displaying picks being made in the Menu Manager menus. This does not affect the display of picks being made in menu bar menus or dialog boxes.	no / yes
visible_message_lines	Sets the default number of visible message lines in the Pro/ENGINEER message area.	
vrml_anchor_url	Enables you to place an anchor on a specified VRML component during the export to VRML. The keywords are optional.	
vrml_background_color	Yes - Exports a model to VRML with the Pro/ENGINEER background color.	yes / no
vrml_explode_lines	Yes - Exports a model to VRML with explode lines for assembly or assembly process data.	yes / no
vrml_explode_lines	Yes - Exports a model to VRML with explode lines for assembly or assembly process data.	yes / no
vrml_export_resolution	Specifies the number of levels of detail (LODs) in models exported in VRML format.	high / medium / low
vrml_export_version	Controls the VRML version for export.	2.0 / 1.0
vrml_file_duplicate_material	Yes - Ensures that the model components retain their true color. No - In some viewers, component colors might not be consistent.	yes / no



Name	Description	Values
vrml_file_duplicate_material	Yes - Ensures that the model components retain their true color. No - In some viewers, component colors might not be consistent.	yes / no
vrml_multiple_views	All - Exports top-level and lower-level assembly component views into VRML format. None - Does not export component views into VRML format. Top - Exports top-level object view only into VRML format.	none / all / top
vrml_parameters	Controls the export of user parameters. Designated - Export only designated parameters. All - Export all parameters. None - Do not export parameters.	designated / all / none
vrml_simpexp_export	Yes - Specifies direct and in memory export of top-level assembly simplified representation to Pro/FLY-THROUGH package files.	yes / no
warn_for_unlocked_obj	Issues you a warning when you fetch, retrieve or modify a Pro/PDM object that has not been locked.	yes / no
web_browser_history_days	Enter number of days to store history records.	
web_browser_homepage	Enter location of Pro/E browser home page.	
web_disable_js_command	Enter a specific javascript command to be disabled	
web_enable_javascript	Enables or disables Pro/Web.Link in the embedded browser.	on / off
web_enable_subscription	Enable the subscription mechanism for Connections	no / yes
web_link_file_read	Sets Pro/Web.Link permission to read data from the local file system.	yes / no
web_link_file_write	Sets Pro/Web.Link permission to write data to the local file system.	yes / no
web_link_proe_read	Sets Pro/Web.Link permission to read data from a Pro/ENGINEER session.	yes / no
web_link_proe_write	Sets Pro/Web.Link permission to modify data in a Pro/Engineer session.	yes / no
web_link_security_autoprompt	Yes - The Security dialog box is displayed when a Pro/Web.Link function requires security access beyond the current settings.	yes / no
web_max_mail_size	Set the maximum email attachment size in kilobytes.	
weld_ask_xsec_refs	No - Do not ask for X-section references when creating weld features. Yes - Ask for X-section references when creating weld features.	no / yes
weld_color	Defines default weld color. The three decimal values specify (in order) percentage of red, green and blue in the resulting color. For example, 0 0 49 specifies a medium blue. Change in session using Tools>Colors>System.	
weld_dec_places	Sets the default number of decimal places (0-10) to display in weld parameters.	
weld_edge_prep_driven_by	Determines if the edge preparation feature is created in the part or the assembly level.	part / assembly
weld_edge_prep_groove_angle	Specifies an initial default value for the angle cut edge preparation.	
weld_edge_prep_groove_depth	Specifies an initial default value for the edge preparation depth.	
weld_edge_prep_instance	Controls whether a family table instance is created for edge preparation. Yes - Instances of the components that receive edge preparation are created.	yes / no
weld_edge_prep_name_suffix	Specifies the suffix name for the instance that will be created during edge preparation. Part name plus extension makes the instance name.	
weld_edge_prep_root_open	Specifies an initial default value for the root opening edge preparation.	
weld_edge_prep_visibility	Applies only if weld_edge_prep is set to Yes. Generic - Edge preparation features are resumed in the generic and suppressed in the instance. Instance - Edge preparation features are suppressed in the generic and resumed in the instance.	generic / instance
weld_notch_corner_radius	Specifies an initial default value for the radius at weld notch corner.	
weld_notch_height	Specifies an initial default value for the weld notch height.	

Name	Description	Values
weld_notch_radius	Specifies an initial default value for the weld notch radius.	
weld_notch_width	Specifies an initial default value for the weld notch width.	
weld_ui_standard	Specifies standard for welding user interface. ANSI - Use ANSI standards. ISO - Use ISO standards.	iso / ansi
windows_scale	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. Refer to menu_horizontal_hint.	
www_add_aux_frame	Creates additional frame for each process step or for assembly publication. Yes - Auxiliary files aux.html are created for each step in assembly process (in each step00 directory) to be replaced by your .html file. No - Auxiliary files are not created.	no / yes
www_export_geometry_as	Specifies the output format by to one of the values	vrml / cgm / jpg / cgm_vrml / jpg_vrml / cgm_jpg / all
www_multiple_views	All - Writes all views in any component to the appropriate VRML files. Top - Writes only named views that reside in the assembly or process assembly to top-level VRML files. None - Does not write named views to VRML files.	none / all / top
www_tree_location	Specifies the location of the model tree in the browser window. Out - Opens model tree in a separate window. In - Includes model tree in the Web page and removes the Tree checkbox from the control panel.	out / in
x_angle	Sets the default view orientation for models (appears in the Orientation dialog box). The default orientation is the last option, x_angle, y_angle, or orientation, in the configuration file.	
y_angle	Sets the default view orientation for models (appears in the Orientation dialog box). The default orientation is the last option, x_angle, y_angle, or orientation, in the configuration file.	

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