

Drawing Setup File Options

The drawing setup file contains many options, each having multiple settings to control the behavior of the drawing. The default setting for each option is shown in ***bold italics***. Other settings are listed below the default in each option. For the options that use a numeric value, such as 'drawing_text_height', the units of the value are controlled by the option called 'drawing_units'. Pick **File, Properties, Drawing Options** to edit these options.

2d_region_columns_fit_text	<i>no</i> , yes	Determines if Two-D repeat regions autosize to fit the longest text in each column.
allow_3d_dimensions	<i>no</i> , yes	Determines if dimensions are shown in isometric views.
angdim_text_orientation	<i>horizontal</i>	Displays text of angular dimensions horizontally at all times, centered between the leaders (equivalent to the value 'horizontal' for the drawing setup file option 'text_orientation').
	parallel_outside	Displays text parallel to the leader lines, regardless of their orientation (equivalent to the value 'parallel' for the drawing setup file option 'text_orientation').
	horizontal_outside	Displays text horizontally outside the dimension.
	parallel_above	Displays text parallel to the dimension arc, but above it.
	parallel_fully_outside	Displays text of angular dimensions (with a plus/minus tolerance) parallel to the leader lines.
asme_dtm_on_dia_dim_gtol	<i>on_gtol</i> , on_dim	Controls the placement of a set datum attached to a diameter dimension. If set to 'on_dim', the set datum attaches to the diameter dimension in accordance with the ASME standard. If set to 'on_gtol', places it on the gtol.
associative_dimensioning	<i>yes</i> , no	Associates draft dimensions to draft entities. The system associates only dimensions that you create while you have this set to 'yes'.
axis_interior_clipping	<i>no</i> , yes	When set to 'no', displays axes in a drawing according to the ANSI Y14.2M standard. When set to 'yes', you can adjust each axis individually by clipping and moving.
axis_line_offset	<i>0.100000</i> , value	Sets the default distance that a linear axis extends beyond its associated feature.
blank_zero_tolerance	<i>no</i> , yes	Controls the display of a plus or minus tolerance value. If set to 'yes', does not display a plus or minus tolerance value if you set the tolerance value to zero.
broken_view_offset	<i>1.000000</i> , value	Sets the offset distance between the two halves of a broken view.

chamfer_45deg_leader_style	<i>std_asme_ansi</i> std_din std_iso std_jis	Controls the leader type of a chamfer dimension without affecting the text.
circle_axis_offset	<i>0.100000</i> , value	Sets the default distance that a circular cross-hair axis extends beyond the circular edge.
clip_diam_dimensions	<i>yes</i> , no	Controls the display of diameter dimensions in detail views. When set to yes, diameter dimensions that fall outside the view will be clipped.
clip_dim_arrow_style	<i>double_arrow</i> arrowhead dot filled_dot arrow slash integral box filled_box	Controls the arrow style of clipped dimensions.
clip_dimensions	<i>yes</i> , no	Controls the display of dimensions in a detailed view. When set to 'yes', the system does not display dimensions completely outside of a detailed view boundary, and the system shows dimensions that cross a detailed boundary with a special double arrow. When set to 'no', the system displays all dimensions.
create_area_unfold_segmented	<i>yes</i> , no	Displays the view in segments corresponding to the straight segments of the cross-section. To draw view borders between view segments, set 'show_total_unfold_seam' to 'yes'. This option only affects <i>new</i> views. When set to 'no', the system does not 'segment' the cross section view.
crosssec_arrow_length	<i>0.187500</i> , value	Sets the length of the cross-section cutting plane arrows.
crosssec_arrow_style	<i>tail_online</i> head_online	Sets the display style of cross-section arrows.
crosssec_arrow_width	<i>0.062500</i> , value	Sets the width of cross-section cutting plane arrows.
crosssec_text_place	<i>after_head</i> before_tail above_tail above_line no_text	Sets the location of cross-section text. When set to 'no_text', the system does not display any cross-section text.
crosssec_type	<i>old_style</i> new_style	Controls the display of cross section views.

cutting_line	<i>std_ansi</i> std_din std_iso std_jis std_ansi_dashed std_alterate	Controls display of cutting line. When set to 'std_ansi', uses the ANSI standard for cutting lines. When set to 'std_ansi_dashed', uses dashed lines. Otherwise, uses the DIN standard cutting line. Displays its thickened portion in white, and displays its thin portion in gray. If set to 'std_jis_alterate', and the drawing setup file option 'cutting_line_segment' is set, displays view arrows as follows: The thickened portion of the cutting line forms an angle and the system displays it in blue. Displays the connecting portions of the cutting line segment between thickened segments in yellow. Displays arrow portions in white. If you set the drawing setup file option 'cutting_line_segment' to '0', displays the entire cutting line as a dashed yellow line. If the length of a cutting line segment is too large, displays the entire cutting line in blue.
cutting_line_adapt	<i>no</i> , yes	Controls display of line fonts used to show cross-sectional arrows. When set to 'yes', all line fonts are display adaptively, beginning in the middle of a complete line segment and ending in the middle of a complete line segment.
cutting_line_segment	<i>0.000000</i> , value	Specifies the length in drawing units of the thickened portion of a non-ANSI cutting line. If set to '0', the length of the cutting line segment is 0.
dash_supp_dims_in_region	<i>yes</i> , no	Controls display of dimension values in Pro/REPORT table repeat regions. When set to 'no', displays the values in Pro/REPORT table repeat regions. When set to 'yes', suppresses the dimension and displays a dash instead.
datum_point_shape	<i>cross</i> dot circle triangle square	Controls display of datum points.
datum_point_size	<i>0.312500</i> , value	Controls size of model datum points and sketched two-dimensional points. The system does not use the units of the drawing or model; it always displays the point size in inches.
decimal_marker	<i>comma_for_metric_dual</i> period comma	Uses a period in the primary dimensions and a comma in secondary dimensions to mark the decimal point when using dual dimensions. Uses a period to mark the decimal point in all dimensions. Uses a comma to mark the decimal point in all dimensions.
default_dim_elbows	<i>yes</i> , no	Controls display of dimension elbows. If set to 'yes', dimensions display with elbows.
default_font	<i>font</i> , font name	Sets default text fonts as those fonts listed in the specified font directory or folder. Do not include the '.ndx' extension.

default_pipe_bend_note	<i>no</i> , value	Controls display of pipe bend notes in drawings. If set as text within quotation marks, uses that value when creating bend notes. Text may include parameters such as '&bend_name:att_pipe_bend' and '&bend_tol:att_pipe_bend'. If set as a directory path, references a previously created note saved as a file.
def_bom_balloons_attachment	<i>edge</i> surface	Sets the default attachment for BOM balloons.
def_bom_balloons_edge_att_sym	<i>arrowhead</i> dot filled_dot no_arrow slash integral box filled_box	Sets the default arrow style for report driven BOM balloons.
def_bom_balloons_snap_lines	<i>no</i> , yes	Determines if snap lines are automatically created when creating BOM balloons.
def_bom_balloons_stagger	<i>no</i> , yes	Determines if BOM balloons are automatically staggered when creating BOM balloons.
def_bom_balloons_stagger_value	<i>0.600000</i> , value	Determines the stagger spacing for BOM balloons.
def_bom_balloons_surf_att_sym	arrowhead dot filled_dot no_arrow slash <i>integral</i> box filled_box	Sets the default arrow style for report driven BOM balloons.
def_bom_balloons_view_offset	<i>0.800000</i> , value	Determines the default distance from the view border to BOM balloons.
def_view_text_height	<i>0.000000</i> , value	Sets the height of text in view names used in view notes and in arrows in cross-sectional and projection detail views.
def_view_text_thickness	<i>0.000000</i> , value	Sets default thickness for new text in view names used in view notes and in arrows in newly created cross-sectional and projection detail views.
def_xhatch_break_around_text	<i>no</i> , yes	Determines if cross hatching breaks around dimensions.
def_xhatch_break_margin_size	<i>0.150000</i> , value	Determines the distance between the cross hatch lines and the dimension text.

detail_circle_line_style	<i>solidfont</i> dotfont ctrlfont phantomfont dashfont	Sets line font for circles indicating a detailed view.
detail_circle_note_text	<i>default</i>	Sets the text used in detail view callout notes.
detail_view_boundary_type	<i>circle</i> ellipse h/v_ellipse spline asme_94_circle	Determines the type of boundary used to callout detail views.
detail_view_circle	<i>on</i> , off	Sets display of a circle drawn about the portion of a model that is detailed by a detailed view.
detail_view_scale_factor	<i>2.000000</i> , value	Sets the initial scale for detailed views. This number is multiplied by the scale of the parent view to set the initial scale of detailed views.
dim_box_dot_style	<i>default</i> filled hollow	Determines the style of boxes and dots when used for leader ends of linear dimensions. When set to 'default', the system uses the 'draw_arrow_style' option.
dim_fraction_format	<i>default</i> std aisc	When set to 'default', the system uses the config.pro option called 'dim_fraction_format'. When set to 'std', the system ignores the config.pro option and uses the PTC standard for fractional dims. When set to 'aisc', the system uses the AISC standard.
dim_leader_length	<i>0.500000</i> , value	Sets length of dimension leader line when leader arrows are outside of witness lines.
dim_text_gap	<i>0.500000</i> , factor	Controls distance between dimension text and dimension leader line and represents the ratio between gap size and text height. For diameter dimensions, if 'text_orientation' is set to 'parallel_diam_horiz', 'dim_text_gap' controls the extension of an elbow beyond the text.
display_tol_by_1000	<i>no</i> , yes	When set to yes, linear tolerances are displayed multiplied by 1000.
draft_scale	<i>1.000000</i> , value	Determines value of draft dimensions relative to actual length of draft entity on drawing.
draw_ang_units	<i>ang_deg</i> ang_min ang_sec	Sets display of angular dimensions in a drawing. If set to 'ang_deg', only decimal degrees are displayed. When set to 'ang_min', the system displays degrees and decimal minutes. When set to 'ang_sec', the system displays degrees, minutes, and decimal seconds.

draw_ang_unit_trail_zeros	<i>yes</i> , no	Controls display of angular dimensions. If set to 'yes', removes trailing zeros (in adherence to ANSI standards) when showing angular dimensions in degrees/minutes/seconds format. If set to 'no', does not display trailing zeros in angular dimensions or tolerances.
draw_arrow_length	<i>0.187500</i> , value	Sets length of leader line arrows.
draw_arrow_style	<i>closed</i> open filled	Sets style of all arrows.
draw_arrow_width	<i>0.062500</i> , value	Sets width of leader line arrows. Drives these other drawing setup file options: draw_attach_sym_height, draw_attach_sym_width, and draw_dot_diameter'
draw_attach_sym_height	<i>default</i> , value	Sets height of leader line slashes, integral signs, and boxes. If set to 'default', uses value set for 'draw_arrow_width'.
draw_attach_sym_width	<i>default</i> , value	Sets width of leader line slashes, integral signs, and boxes. If set to 'default', uses value set for 'draw_arrow_width'.
draw_cosms_in_area_xsec	<i>no</i> , yes	Controls display of cosmetic sketches and datum curve features that lie in the cutting plane in planar area cross-sectional views. If set to 'yes', shows all cosmetic sketches and datum curve features that lie in the cutting plane. If set to 'no', does not show them.
draw_dot_diameter	<i>default</i> , value	Sets diameter of leader line dots. If set to 'default', uses value set for 'draw_arrow_width'.
draw_layer_overrides_model	<i>no</i> , yes	Directs drawing layer display setting to determine the setting of drawing model layers with the same name. If set to 'yes', implicitly includes drawing model layers in drawing layers with the same name for purposes of setting the display. If set to 'no', ignores nondrawing layers when the display status of layers is set in the drawing model.
drawing_text_height	<i>0.15625</i> , value	Sets default text height for all text in the drawing.
drawing_units	<i>inch</i> foot mm cm m	Sets units for all drawing setup file parameters.
dual_digits_diff	<i>-1</i> , value	Controls number of digits to the right of the decimal that the secondary dimension differs from primary dimension in dual dimensioning.
dual_dimension_brackets	<i>yes</i> , no	Controls display of brackets with dimension units. This option works only when you are using 'dual_dimensioning'. If set to 'yes', displays dimension units that occur second in brackets; if set to 'no', does not display brackets.

dual_dimensioning	<i>no</i>	Controls format of dimension display. If set to 'no', displays a single value for dimensions.
	primary[secondary]	Displays dimensions with primary units (established by the model) on top and secondary units on the bottom.
	secondary[primary]	Displays dimensions with secondary units on top and primary units on the bottom. This is not recommended.
	secondary	Displays only the secondary dimensions of the drawing, as if they were primary. This is not recommended.
dual_metric_dim_show_fractions	<i>no</i> , yes	Control the metric portion of dual dimensions when the primary units are using fractions.
dual_secondary_units	<i>mm</i>	Sets units for the display of secondary dimensions.
	inch	
	foot	
	cm	
	m	
gtol_datums	<i>std_ansi</i>	Sets drafting standard followed for displaying reference datums in drawings. Display affects both axes and datum planes, and display of reference part datums.
	std_ansi_mm	
	std_iso	
	std_jis, std_din	
	std_iso_jis, std_asme	
	std_ansi_dashed	
gtol_dim_placement	<i>on_bottom</i>	Determines the location of a feature control frame of a geometric tolerance when attached to a dimension symbol that contains additional text. 'On_bottom' places the geometric tolerance at the bottom of the dimension symbol, beneath any additional lines of text. When set to 'under_value', the system places the geometric tolerance immediately below the dimension value and above any additional lines of text.
	under_value	
half_view_line	<i>solid</i>	Draws solid lines where material is present.
	symmetry	Draws a centerline extending beyond the part and acting as a break line.
	none	Draws the object a small distance past the symmetry line. You must select an offset datum to create the half view.
harn_tang_line_display	<i>no</i> , yes	Controls display of internal segment portions of cables.
hidden_tangent_edges	<i>default</i>	Controls display of hidden tangent edges in drawing views. If set to 'dimmed', plots hidden tangent edges in a view using Pen 7. Lines appear dashed in same color as dimmed visible tangent edges. However, you must select Hidden Line or No Hidden from the Display Style list in the Environment dialog box. If set to 'erased', removes all hidden tangent edges automatically from screen and plot.
	dimmed	
	erased	

hlr_for_pipe_solid_cl	<i>no</i> , yes	Controls display of pipe centerlines. If set to 'yes', hidden line removal affects pipe centerlines. If set to 'no', it does not. Operates only on pipes created in Pro/PIPING, not on pipe features in a part.
hlr_for_threads	<i>yes</i> , no	Controls display of threads in a drawing depending on whether it complies with the ISO or ANSI standard (set by the 'thread_standard' option). If set to 'yes', thread edges meet ANSI or ISO standard for Hidden Line display.
ignore_model_layer_status	<i>yes</i> , no	Controls whether the system considers layer status in models. If set to 'yes', ignores changes to all layer status in the models of the drawing made in another mode.
iso_ordinate_delta	<i>no</i> , yes	Improves display of offset between an ISO-ordinate dimension line and witness line, referred to as the 'witness line delta'. If set to 'no', the system does not display the offset exactly in accordance with the specified value (it is 'off' by about 2 millimeters). When set to 'yes', the system displays the offset correctly, according to value specified for the drawing setup file option 'witness_line_delta'.
leader_elbow_length	<i>0.250000</i> , value	Determines length of leader line elbows.
leader_extension_font	fontname	Determines the line font of drawing leaders.
lead_trail_zeros	<i>std_default</i>	Controls use of leading and trailing zeros in a dimension, enabling independent control over the dimension value display for primary and secondary units in the drawing. If set to 'std_default', the system displays all dimension values according to the units of the model. Metric parts follow the 'std_metric' rule (see below), while inch parts follow the 'std_english' rule.
	std_metric	When set to 'std_metric', the system displays all values with leading zeros and without trailing zeros, regardless of the units of the model.
	std_english	When set to 'std_english', the system displays all values without leading zeros, with trailing zeros, regardless of the units of the model.
	both	When set to 'both', the system show all values with both leading and trailing zeros. When using dual dimensioning, these values can be mixed. Example: std_english[both]
lead_trail_zeros_scope	<i>dims</i> , all	When set to 'dims', the system applies the 'lead_trail_zeros' option to dimensions only. When set to 'all', the system applies the 'lead_trail_zeros' option to dimensions, parameters, parametric notes, view scale notes, tables, symbols, and cosmetic thread notes.

line_style_standard	<i>std_ansi</i> std_iso std_jis std_din	Controls text color in drawings. Unless set to 'std_ansi', displays all drawing text in blue, and displays boundary of detailed views in yellow.
location_radius	<i>default(2.)</i> 0.0 value	Modifies radius of nodes indicating location, improving their visibility, particularly when printing drawings. If set to 'default', sets radius as 2 drawing units. If set to '0.0', displays location nodes, but does not print them. There is no maximum value for this setting.
max_balloon_radius	<i>0.000000</i> , value	Sets the maximum allowable balloon radius. If set to '0', balloon radius depends only on text size.
mesh_surface_lines	<i>on</i> , off	Controls display of blue surface mesh lines.
min_balloon_radius	<i>0.000000</i> , value	Sets minimum allowable balloon radius. If set to '0', balloon radius depends only on text size.
min_dist_between_bom_balloons	<i>0.800000</i> , value	Sets the default spacing between BOM balloons.
model_digits_in_region	<i>yes</i> , no	Controls display of number of digits in two-dimensional repeat regions. If set to 'yes', two-dimensional repeat regions reflect the number of digits of part and assembly model dimensions.
model_display_for_new_views	<i>default</i> wireframe hidden_line no_hidden save_environment	Controls the view display for new views. 'Default' uses the environment setting for display.
model_grid_balloon_display	<i>yes</i> , no	Controls the display of model grid balloons.
model_grid_balloon_size	<i>0.200000</i> , value	Specifies default radius of balloons shown with the model grid in a drawing.
model_grid_neg_prefix	<i>-</i> , string	Controls prefix of negative values shown in model grid balloons.
model_grid_num_dig_display	<i>0</i> , integer value	Controls number of digits displayed in grid coordinates that appear in grid balloons. Type an integer specifying the number of decimal places, or use the system default (0) to display coordinates as integers.
model_grid_offset	<i>default</i> , value	Controls offset of new model grid balloons from the drawing view. If set to 'default', offsets new model grid balloons from the drawing view by twice the current model grid spacing. If set to a value, offsets balloons by that number of inches (not drawing units) from the view.
model_grid_text_orientation	<i>horizontal</i> parallel	Control the orientation of the model grid text.

model_grid_text_position	<i>centered</i> above below	Controls the position of the model grid text relative to the grid line when the text orientation is set to 'parallel'.
new_iso_set_datums	<i>yes</i> , no	Controls display of set datums. If set to 'yes', displays set draft datums in accordance with the ISO standard.
node_radius	<i>default</i> , value	Controls display of nodes in symbols. If set to 'default', system specifies radius of nodes. If the setting is so small that the nodes do not appear, the system uses the default setting. There is no maximum value for this setting.
ord_dim_standard	<i>std_ansi</i> std_iso std_jis std_din	Sets the standard according to the display of ordinate dimensions. If set to 'std_ansi', shows dimensions without a connecting line. When set to any of the other options, the system places related ordinate dimensions along the connecting line that is perpendicular to the baseline and starts with a circle. Each segment of the connecting line ends with an arrow. Displays arrows and the circle filled or open, according to the current setting of the drawing setup file option 'draw_arrow_style'. Note that when witness lines are interconnected, moving any of the related dimensions moves all of them.
orddim_text_orientation	<i>parallel</i> horizontal	Displays dimension text parallel to the leader lines. Display ordinate dimension text horizontally, parallel to the bottom of the drawing sheet.
parallel_dim_placement	<i>above</i> below	Dimension value is displayed above the leader line when the 'text_orientation' option is set to 'parallel'. This option does not apply to dual dimensions. When set to 'below', the system displays the dimension value below the leader line.
pipe_pt_line_style	default	Controls the font used to display bend intersection points on a piping drawing.
pipe_pt_shape	<i>cross</i> dot circle triangle square	Controls shape of theoretical bend intersection points in a piping drawing.
pipe_pt_size	<i>default</i> , value	Controls size of theoretical bend intersection points in a piping drawing.
pos_loc_format	%s%x%y, %r	This string controls the format of the &pos_loc parameter shown in notes and report tables. %s shows the sheet number. %x shows the horizontal position of the note. %y shows the vertical position, and %r ends the string.
projection_type	<i>third_angle</i> first_angle	Determines method for creating projection views.

radial_dimension_display	<i>std_asme</i> std_iso std_jis	Controls the display of radial dimensions.
radial_pattern_axis_circle	<i>no</i> , yes	Sets display mode for axes of rotation that are perpendicular to the screen in radial pattern features. If set to 'no', displays axis lines. If set to 'yes', a circular shared axis appears, and axis lines pass through the center of a rotational pattern.
ref_des_display	yes, <i>no</i> , default	Controls display of reference designators in a drawing of a cabling assembly. If set to 'default', selects the Reference Designators checkbox in the Environment dialog box.
reference_bom_balloon_text	<i>"default"</i> , string	Sets the text used in reference BOM balloons.
remove_cosms_from_xsecs	<i>total</i> , all, none	Controls display of datum curves, threads, cosmetic feature entities, and cosmetic cross-hatching in a full cross-sectional view. If set to 'all', removes datums and cosmetics from all types of cross-sectional views. If set to 'total', removes features located entirely in front of the cutting plane from the cross-sectional view. Displays these features in full only if they intersect the cutting plane. If set to 'none', displays all datum quilts and cosmetic features.
show_cbl_term_in_region	<i>yes</i> , no	Allows use of the report symbols '&asm.mbr.name' and '&asm.mbr.type' to show terminators in Pro/REPORT tables for cable assemblies having connectors with terminator parameters. If set to 'yes' (you must set the attribute Cable Info for the repeat region), shows terminators. When creating new drawings, the default value is 'yes'. For existing drawings, the default value is 'no'.
show_pipe_theor_cl_pts	<i>bend_cl</i> theor_cl both	Controls display of centerlines and theoretical intersection points in piping drawings. When set to 'bend_cl', shows centerlines with bends only. Shows only centerlines with theoretical bend intersection points. Shows both bends and theoretical intersection points.
show_quilts_in_total_xsecs	<i>no</i> , yes	Controls the display of surface quilts in total cross section views.
show_sym_of_suppressed_weld	<i>no</i> , yes	Controls the display of weld symbols for suppressed welds when using Pro/WELDING.
show_total_unfold_seam	<i>yes</i> , no	Display the seams (the edges of the cutting plane) in total unfolded cross-sectional views. When set to 'no', the system blanks the seams (the edges of the cutting plane) in total unfolded cross-sectional views.
shrinkage_value_display	<i>percent_shrink</i> final_value	Displays dimension shrinkage in percentages. Displays dimension shrinkage as the final value of the dimension.

sort_method_in_region	<i>delimited</i> string_only trailing_numbers pre_2001	Determines how repeat regions are sorted by default.
sym_flip_rotated_text	yes, <i>no</i>	Flips any text in a Rotate Text symbol that is upside down, making it right-side up. If set to 'yes' and the symbol orientation is +/- 90 degrees, flips the text, rotating it along with the symbol (for an illustration, see Manipulating Instances).
symmetric_tol_display_standard	<i>std_asme</i> std_iso std_din	Controls how symmetric tolerances are displayed.
tan_edge_display_for_new_views	<i>default</i> tan_solid no_disp_tan tan_ctrln tan_phantom tan_dimmed save_environment	Controls the display of tangent edges for new views. 'Default' uses the environment setting for display.
text_orientation	<i>horizontal</i> parallel parallel_diam_horiz	Controls orientation of dimension text in the drawing. Displays all dimension text horizontally. Displays text parallel to the dimension leader line. Displays all dimensions except diameter dimensions parallel to their leaders; displays only diameter dimensions horizontally. The elbow of a diameter dimension always extends through to the end of the text. To control the elbow extension beyond the text, use the drawing setup file option 'dim_text_gap'. The drawing setup file option 'angdim_text_orientation' (not 'text_orientation') controls the display of angular dimensions.
text_thickness	<i>0.000000</i> 0 < value < 0.5	Sets default text thickness for new text and existing text whose thickness has not been modified. Enter a value between 0 and 0.5.
text_width_factor	<i>0.800000</i> 0.25 < value < 8	Sets default ratio between the text width and text height. The system maintains this ratio until you change the width using the Text Style dialog box. Enter a value between 0.25 and 8.

thread_standard	<i>std_ansi</i>	Controls display of threaded hole with an axis (perpendicular to the screen as an arc (ISO standard) or as a circle (ANSI standard)).
	std_ansi_imp	If set to 'std_ansi_imp' or 'std_iso_imp', does not display hidden thread lines when you select No Hidden Line from the Display Style list in the Pro/ENGINEER Environment dialog box. When you select Hidden Line, displays thread lines as leader lines (yellow).
	std_iso_imp_assy	If set to 'std_iso_imp_assy', displays threads in cross-sectional assembly drawings in accordance with the ISO 6410 standard.
	std_ansi_imp_assy	If set to 'std_ansi_imp_assy', displays them in accordance with the ANSI standard. The 'std_iso' and 'std_ansi' values are valid for drawings created before Release 15.0.
tol_display	<i>no</i> , yes	Do not display dimensions with their tolerances. You cannot access the 'Tolerances' button in the 'Environment' dialog box when this option is set to no. When set to 'yes', the system display dimensions with their tolerances.
tol_text_height_factor	<i>standard</i> , factor	Sets default ratio between the tolerance text height and dimension text height, when showing the tolerance in 'plus-minus' format. If set to 'standard', the system uses 1 for the ANSI standard and .6 for the ISO standard.
tol_text_width_factor	<i>standard</i> , factor	Sets default factor to maintain a proportion between the tolerance text width and dimension text width, when showing the tolerance in 'plus-minus' format. If set to 'standard', the system uses .8 for the ANSI standard and .6 for the ISO standard.
use_major_units	<i>no</i> , yes, default	Controls fractional dimension display. When set to 'default', the system uses the config.pro option called 'use_major_units'.
view_note	<i>std_ansi</i>	Creates view-related notes with the words 'SECTION', 'DETAIL', and 'SEE DETAIL' included.
	std_din	Creates view-related notes with the words 'SECTION', 'DETAIL', and 'SEE DETAIL' omitted.
	std_iso	Same as std_ansi
	std_jis	Same as std_ansi
view_scale_denominator	<i>0</i> integer	Expresses the view scale value in decimal format. Set the denominator for the view scale.
view_scale_format	<i>decimal</i> fractional ratio_colon	Expresses a scale as a decimal or fractional value. If set to 'ratio_colon', the system displays the view scale values as a ratio. For example, instead of a view scale of 0.5, display the view scale as 1:2. Be sure to set the 'view_scale_denominator' option appropriately.
weld_light_xsec	<i>no</i> , yes	Determines if light weight weld cross sections are shown.
weld_solid_xsec	<i>no</i> , yes	Displays weld features as solids in cross section views.

weld_symbol_standard	<i>std_ansi</i> <i>std_iso</i>	Displays weld symbols in a drawing according to the ANSI or ISO standard.
witness_line_delta	<i>0.125000</i> , value	Sets the extension of the witness line beyond the dimension leader arrows.
witness_line_offset	<i>0.062500</i> , value	Sets offset between a dimension line and object being dimensioned. This gap may be visible only when you plot a drawing. To see the effect, use the screen plot. Also controls the size of the break at the intersection of witness lines, when you use the 'Dimension' type break.
yes_no_parameter_display	<i>true_false</i> <i>yes_no</i>	Controls display of 'yes/no' parameters in drawing notes and tables. When set to 'yes_no', parameters can have a 'yes' or 'no' value in drawing notes. When set to 'true_false', they can have a 'true' or 'false' value.
zero_quantity_cell_format	<i>empty</i> - 0	Controls the display of repeat region cells with a quantity of zero.