SOLIDWORKS TOOLBARS AND ICONS

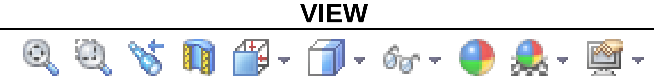
STANDARD

	NEW	Creates a new document
	OPEN	Opens an existing document
	SAVE	Saves the active document
	PRINT	Prints the active document
	UNDO	Reverses the last action
	SELECT	Select sketch entities, edges, vertices, components and so on
	REBUILD	Rebuilds the part/assembly/drawing
	FILE PROPERTIES	Shows the summary information of the active document
	OPTIONS	Change option settings for SolidWorks

SPLINE TOOLS

	ADD TANGENCY CONTROL	Adds a tangency control handle that you drag along the spline, position, then use to control the tangency at that point
	ADD CURVATURE CONTROL	Adds a curvature control handle that you drag along the spline, position, then use to control the spline shape at that point
	INSERT SPLINE POINT	Adds a point to a spline. You can drag points to reshape splines and add dimensions between spline points
	SIMPLIFY SPLINE	Reduces the number of points in a selected spline, which improves performance in models with complex spline curves
	FIT SPLINE	Adds a spline based on selected sketch entities and edges
	SHOW SPLINE HANDLES	Displays all handles of a selected spline. You drag handles to reshape a spline
	SHOW INFLECTION POINTS	Displays all points where the concavity of a selected spline changes
	SHOW MINIMUM RADIUS OF CURVATURE	Displays the measurement of the smallest radius in a selected spline
	SHOW CURVATURE	Displays scalable curvature combs that visually enhance the curves of a selected spline

VIEW



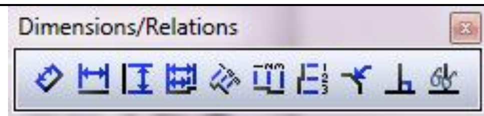
	ZOOM TO FIT	Zooms the model to fit the window
	ZOOM TO AREA	Zooms to the area you select with a

		bounding box
	PREVIOUS VIEW	Displays the previous view
	SECTION VIEW	Displays a cutaway of a part or assembly using one or more cross section planes
	VIEW ORIENTATION	Changes the current view orientation or number of viewpoirts.
	DISPLAY STYLE	Changes the display style for the active view.
	HIDE/SHOW ITEMS	Changes the visibility of items in the graphics area.
	EDIT APPEARANCE	Edit the appearance of entities in the model.
	APPLY SCENE	Cycles through or applies a specific scene.
	VIEW SETTINGS	Toggle various view settings such as RealView, Shadows, and Perspective.

SKETCH

	Sketch	Creates a new sketch, or edits an existing sketch.
	3D SKETCH	Adds a new 3D sketch, or edits an existing 3D sketch
	SMART DIMENSION	Creates a dimension for one or more selected entities
	LINE	Sketches a line
	RECTANGLE	Sketches a rectangle
	CIRCLE	Sketches a circle.
	CENTERPOINT ARC	Sketches a center point arc.
	TANGENT ARC	Sketches an arc tangent to a sketch entity.
	3 POINT ARC	Sketches a 3 point arc.
	SKETCH FILLET	Rounds the corner at the intersection of two sketch entities, creating a tangent arc
	CENTERLINE	Sketches a centerline.
	SPLINE	Sketches a spline
	POINT	Sketches a point
	PLANE	Inserts a plane into the 3D sketch
	MIRROR ENTITIES	Mirrors selected entities about a centerline
	CONVERT ENTITIES	Converts selected model edges or sketch entities into sketch segments
	OFFSET ENTITIES	Adds sketch entities by offsetting faces, edges, or sketch entities a specified distance
	TRIM ENTITIES	Trims or extends a sketch entity to be coincident to another, or deletes a sketch entity
	CONSTRUCTION GEOMETRY	Toggles sketch entities between construction geometry and normal sketch geometry
	MOVE ENTITIES	Moves sketch entities and annotations

DIMENSIONS / RELATIONS

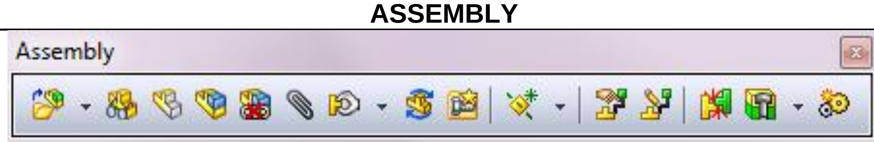


	SMART DIMENSION	Creates a dimension for one or more selected entities
	HORIZONTAL DIMENSION	Creates a horizontal dimension between selected entities
	VERTICAL DIMENSION	Creates a vertical dimension between selected entities
	BASELINE DIMENSION	Creates a reference dimension between selected entities
	ORDINATE DIMENSION	Creates a set of dimensions measured from a zero ordinate in a drawing or sketch
	HORIZONTAL ORDINATE DIMENSION	Creates horizontal ordinate dimensions in a drawing or sketch, measured horizontally from the first selected entity
	VERTICAL ORDINATE DIMENSION	Creates vertical ordinate dimensions in a drawing or sketch, measured vertically from the first selected entity
	CHAMFER DIMENSION	Creates dimensions of chamfers in drawings
	ADD RELATION	Controls the size or position of entities with constraints such as concentric or vertical
	DISPLAY/DELETE RELATIONS	Displays and deletes geometric relations

MACRO

	RUN MACRO	Runs an already record macro
	STOP MACRO	Stops the recording of a macro
	RECORD\PAUSE MACRO	Records (or pauses recording of) actions to create a macro
	NEW MACRO	Launches the macro editor and begins editing a new macro
	EDIT MACRO	Opens a macro file for editing

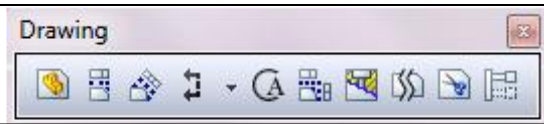
ASSEMBLY



	INSERT COMPONENTS	Adds an existing part or sub-assembly to the assembly
	HIDE/SHOW COMPONENTS	Hides or shows components
	CHANGE SUPPRESSION STATE	Suppresses or resolves components. Suppressed components are not in memory or visible
	EDIT COMPONENT	Toggles between editing a part or a sub-assembly and the main assembly
	NO EXTERNAL REFERENCES	External references will not be created when creating or editing features in context
	MATE	Position two components relative to one another

	MOVE COMPONENT	Moves a component within the degrees of freedom defined by its mates
	ROTATE COMPONENT	Rotates a component within the degrees of freedom defined by its mates
	SMART FASTENERS	Adds fasteners to the assembly using the SolidWorks Toolbox library of standard hardware
	EXPLODED VIEW	Separates the component into an exploded view
	EXPLODE LINE SKETCH	Adds or edits a 3D sketch showing the relationship between exploded components
	INTERFERENCE DETECTION	Detects any interference between components
	ASSEMBLY FEATURES	Creates various assembly features.
	NEW MOTION STUDY	Inserts new motion study.

DRAWING



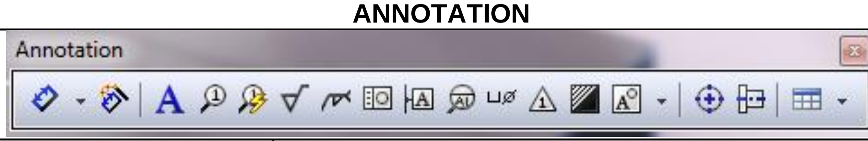
	MODEL VIEW	Adds an orthogonal or named view based on an existing part or assembly
	PROJECTED VIEW	Adds a projected view by unfolding a new view from an existing view
	AUXILLARY VIEW	Adds a view by unfolding a new view from a linear entity (edge, sketch entity, and so on)
	SECTION VIEW	Adds a section view by cutting the parent view with a section line
	ALIGNED SECTION VIEW	Adds an aligned section view using two line connected at an edge
	DETAIL VIEW	Adds a detail view to show a portion of a view, usually at an enlarged scale
	STANDARD 3 VIEW	Adds three standard, orthogonal views. The type and orientation of the views can be first or third angle
	BROKEN-OUT SECTION	Adds a broken-out section to an existing view exposing inner details of a model
	BREAK	Add break lines to the selected view
	CROP VIEW	Crops an existing view to show only a portion of the view
	ALTERNATE POSITION VIEW	Adds a view displaying a configuration of a model superimposed on another configuration of the model

FEATURES

	EXTRUDED BOSS/BASE	Extrudes a sketch or selected sketch contours in one or two directions to create a solid feature
	EXTRUDED CUT	Cuts a solid model by extruding a sketched profile in one or two directions
	REVOLVED	Revolves a sketch or selected sketch contours around an axis

	BOSS/BASE	to create a solid feature
	REVOLVED CUT	Cuts a solid model by revolving a sketched profile around an axis
	SWEEP BOSS/BASE	Sweeps a closed profile along an open or closed path to create a solid feature
	LOFTED BOSS/BASE	Adds material between two or more profiles to create a solid feature
	FILLET	Creates a rounded internal or external face along one or more edges in solid or surface feature
	CHAMFER	Creates a bevel feature along an edge, a chain of tangent edges, or a vertex
	RIB	Adds thin-walled support to a solid body
	SHELL	Removes material from a solid body to create a thin-walled feature
	DRAFT	Tapers model faces by a specified angle, using a neutral plane or a parting line
	HOLE WIZARD	Inserts a hole using a pre-defined cross-section
	LINEAR PATTERN	Patterns features, faces, and bodies in one or two linear directions
	CIRCULAR PATTERN	Patterns features, faces, and bodies around an axis
	MIRROR	Mirrors features, faces, and bodies about a face or a plane
	REFERENCE GEOMETRY	Reference Geometry commands
	CURVES	Curve commands

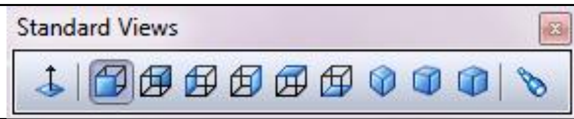
ANNOTATION



	SMART DIMENSION	Creates a dimension for one or more selected entities
	MODEL ITEMS	Imports dimensions, annotations and reference geometry from the referenced model into the selected view
	NOTE	Adds a note
	BALLOON	Adds a balloon
	AUTOBALLOON	Adds balloons for all components in the selected views
	SURFACE FINISH	Adds a surface finish symbol
	WELD SYMBOL	Adds a weld symbol on a selected entity (face, edge, and so on)
	GEOMETRIC TOLERANCE	Adds a geometric tolerance symbol
	DATUM FEATURE	Adds a datum feature symbol
	DATUM TARGET	Adds a datum target (point or area) and symbol
	HOLE CALLOUT	Adds a hole callout
	REVISION SYMBOL	Insert latest revision symbol
	AREA HATCH / FILL	Adds a crosshatch pattern or solid fill to a model face or a closed sketch profile
	BLOCK	Adds a block, which typically contains drawing items that you use often

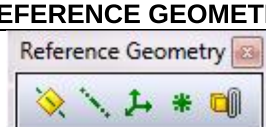
	CENTER MARK	Adds a center mark on a circular edge or sketch entity
	CENTRELINE	Adds centerlines to a view or to selected entities
	TABLES	Table Commands

STANDARD VIEWS



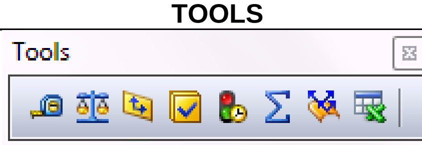
	NORMAL TO	Rotates and zooms the model to the normal to view orientation based on the selected plane, planar face, or feature
	FRONT	Rotates and zooms the model to the front view orientation
	BACK	Rotates and zooms the model to the back view orientation
	LEFT	Rotates and zooms the model to the left view orientation
	RIGHT	Rotates and zooms the model to the right view orientation
	TOP	Rotates and zooms the model to the top view orientation
	BOTTOM	Rotates and zooms the model to the bottom view orientation
	ISOMETRIC	Rotates and zooms the model to the isometric view orientation
	TRIMETRIC	Rotates and zooms the model to the trimetric view orientation
	DIMETRIC	Rotates and zooms the model to the dimetric view orientation
	VIEW ORIENTATION	Displays a dialog box to select standard or user defined views

REFERENCE GEOMETRY



	PLANE	Adds a reference plane
	AXIS	Adds a reference axis
	COORDINATE SYSTEM	Defines a coordinate system for a part or assembly
	POINT	Adds a reference point
	MATE REFERENCE	Specifies entities to use as references for automatic mating using SmartMates

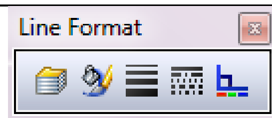
TOOLS



	MEASURE	Calculates the distance between selected items

	MASS PROPERTIES	Calculates the mass properties of the model
	SECTION PROPERTIES	Evaluate section properties for multiple faces and sketches that lie in parallel planes
	CHECK	Checks the model for geometry errors
	STATISTICS	Displays part and assembly statistics such as feature rebuild time and the number of assembly components
	EQUATIONS	Creates mathematical relations between model dimensions, using dimension names as variables
	DEVIATION ANALYSIS	Calculates the angle between faces
	DESIGN TABLE	Inserts or edits a table to build multiple configurations of parts or assemblies

LINE FORMAT



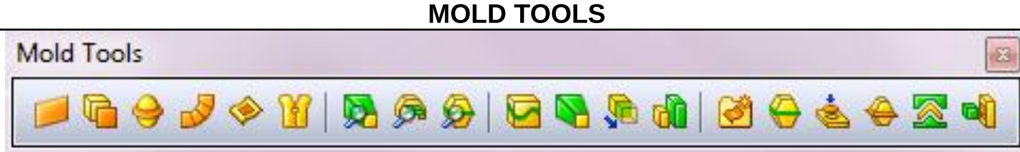
	LAYER PROPERTIES	Creates, edits, or deletes layers. Also, changes the properties and visibility of layers
	LINE COLOR	Changes the color of edges, sketch entities, and many annotation types
	LINE THICKNESS	Changes the thickness of edges and sketch entities
	LINE STYLE	Changes the style of edges and sketch entities
	COLOR DISPLAY MODE	Toggles the color of edges and sketch entities between their layer or line color and the system status colors

SELECTION FILTER

	TOGGLE SELECTION FILTERS	Turns selection filters on and off
	CLEAR ALL FILTERS	Clears all selection filters
	SELECT ALL FILTERS	Selects all selection filters
	INVERT SELECTION	Inverts current selection
	FILTER VERTICES	Allows selection of vertices only
	FILTER EDGES	Allows selection of edges only
	FILTER FACES	Allows selection of faces only
	FILTER SURFACE BODIES	Allows selection of surface bodies only
	FILTER SOLID BODIES	Allows selection of solid bodies only
	FILTER AXES	Allows selection of axes only
	FILTER PLANES	Allows selection of planes only

	FILTER SKETCH POINTS	Allows selection of sketch points only
	FILTER SKETCH SEGMENTS	Allows selection of sketch segments only
	FILTER MIDPOINTS	Allows selection of mid points only
	FILTER CENTER MARKS	Allows selection of center marks only
	FILTER CENTRELINES	Allows selection of centrelines only
	FILTER DIMENSION/HOLE CALLOUTS	Allows selection of dimensions and hole callouts only
	FILTER SURFACE FINISH SYMBOLS	Allows selection of surface finish symbols only
	FILTER GEOMETRIC TOLERANCES	Allows selection of geometric tolerance symbols only
	FILTER NOTES/BALLOONS	Allows selection of notes and balloons only
	FILTER DATUM FEATURES	Allows selection of datum feature symbols only
	FILTER WELD SYMBOLS	Allows selection of weld symbols only
	FILTER WELD BEADS	Allows selection of weld beads only
	FILTER DATUM TARGETS	Allows selection of datum target symbols only
	FILTER COSMETIC THREADS	Allows selection of cosmetic threads only
	FILTER BLOCKS	Allows selection of blocks only
	FILTER DOWEL PIN SYMBOLS	Allows selection of Dowel pin symbols only
	FILTER CONNECTION POINTS	Allows selection of connection points only
	FILTER ROUTING POINTS	Allows selection of routing points only

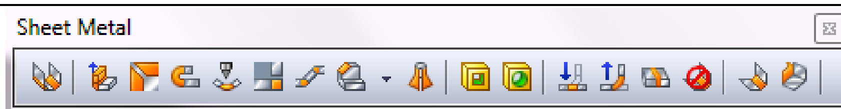
MOLD TOOLS



	PLANAR SURFACE	Creates a planar surface using a sketch or a set of edges
	OFFSET SURFACE	Creates offset surfaces using one or more contiguous faces
	RADIATE SURFACE	Radiate a surface originating from an edge parallel to a plane
	RULED SURFACE	Inserts ruled surfaces from edges
	FILLED SURFACE	Constructs a surface patch within a boundary defined by existing model edges, sketches, or curves
	KNIT SURFACES	Combines two or more adjacent, non-intersecting surfaces together
	DRAFT ANALYSIS	Analyzes draft angle of faces based on a mold pull direction
	UNDERCUT ANALYSIS	Identifies faces that form undercuts
	SPLIT LINE	Projects a sketch to curved or planar faces, creating multiple separate faces

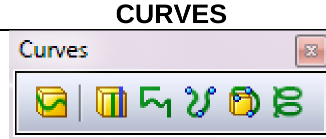
	DRAFT	Tapers model faces by a specified angle, using a neutral plane or a parting line
	MOVE FACE	Move face(s) of a solid
	SCALE	Scale the model by a specified factor
	INSERT MOLD FOLDERS	Insert surface body folders for mold operations
	PARTING LINES	Establishes parting lines to separate core and cavity surfaces
	SHUT-OFF SURFACES	Finds and creates mold shut-off surfaces
	PARTING SURFACES	Creates parting surfaces between core and cavity surfaces
	TOOLING SPLIT	Inserts a Tooling Split feature
	CORE	Extracts core(s) from existing tooling split

SHEET METAL



	BASE-FLANGE/TAB	Creates a sheet metal part or adds a material to an existing sheet metal part
	EDGE FLANGE	Adds a wall to an edge of a sheet metal part
	MITER FLANGE	Adds a series of flanges to one or more edges of a sheet metal part
	HEM	Curls edges of a sheet metal part
	SKETCHED BEND	Adds a bend from a selected sketch in a sheet metal part
	CLOSED CORNER	Extends the face of a sheet metal part
	JOG	Adds two bends from a sketched line in a sheet metal part
	CORNERS	Creates various corner treatments on a sheet metal part
	LOFTED-BEND	Creates a sheet metal part between two sketches using a loft feature
	EXTRUDED CUT	Cuts a solid model by extruding a sketched profile in one or two directions
	SIMPLE HOLE	Creates a cylindrical hole on a planar face
	UNFOLD	Unfolds bends in a sheet metal part
	FOLD	Folds flattened bends in a sheet metal part
	FLATTEN	Shows the flat pattern for the existing sheet metal part
	NO BENDS	Rolls back all bends in the sheet metal part
	INSERT BENDS	Creates a sheet metal part from the existing part
	RIP	Creates a gap between two edges in a sheet metal part

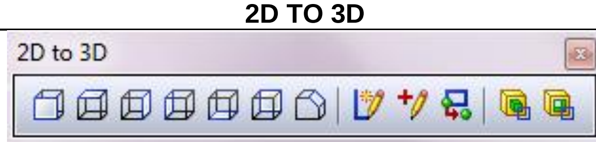
SURFACES		
		
	EXTRUDED SURFACE	Creates an extruded surface
	REVOLVED SURFACE	Creates surface feature by revolving an open or closed path around an axis
	SWEPT SURFACE	Creates surface feature by sweeping an open or closed profile along an open or closed path
	LOFTED SURFACE	Creates a lofted surface between two or more profiles
	PLANAR SURFACE	Creates a planar surface using a sketch or a set of edges
	FILLED SURFACE	Constructs a surface patch within a boundary defined by existing model edges, sketches, or curves
	OFFSET SURFACE	Creates offset surfaces using one or more contiguous faces
	RULED SURFACE	Inserts ruled surfaces from edges
	DELETE FACE	Deletes faces from solid bodies to create surfaces, or deletes from surface bodies
	REPLACE FACE	Replaces faces on a solid or surface body
	KNIT SURFACES	Combines two or more adjacent, non-intersecting surfaces together
	EXTEND SURFACE	Extends the edge, multiple edges or the face on a surface, based on end conditions and extension type
	TRIM SURFACE	Trims a surface where one surface intersects with another surface, a plane, or a sketch
	UNTRIM SURFACE	Patches surface holes and external edges by extending the surfaces
	FILLET	Creates a rounded internal or external face along one or more edges in solid or surface feature
	REFERENCE GEOMETRY	Reference Geometry commands
	CURVES	Curve commands

CURVES		
		
	SPLIT LINE	Projects a sketch to curved or planar faces, creating multiple separate faces
	PROJECT CURVE	Projects a sketched curve onto a face or sketch
	COMPOSITE CURVE	Combines selected edges, curves, and sketches into a single curve
	CURVE THROUGH XYZ POINTS	Adds a curve through X, Y, and Z coordinates that you define
	CURVE THROUGH REFERENCE POINTS	Adds a curve through selected reference points located on one or more planes
	HELIX	Adds a helix or spiral curve from a sketched circle

ALIGN

	GROUP	Creates a group from the selected items
	UNGROUP	Delete the grouping between these items
	ALIGN COLLINEAR/RADIAL	Aligns and groups selected dimensions along a line or an arc
	ALIGN PARALLEL/CONCENTRIC	Aligns and groups selected dimensions at a uniform distance from each other
	ALIGN LEFT	Aligns the left edges of the selected annotations
	ALIGN RIGHT	Aligns the right edges of the selected annotations
	ALIGN TOP	Aligns the top edges of the selected annotations
	ALIGN BOTTOM	Aligns the bottom edges of the selected annotations
	ALIGN HORIZONTAL	Aligns the centers of the selected annotations horizontally
	ALIGN VERTICAL	Aligns the centers of the selected annotations vertically
	ALIGN BETWEEN LINES	Aligns the centers of the selected annotations between the nearest selected horizontal or vertical lines
	SPACE EVENLY ACROSS	Evenly spaces the selected annotations horizontally
	SPACE EVENLY DOWN	Evenly spaces the selected annotations vertically
	SPACE TIGHTLY ACROSS	Tightly spaces the selected annotations horizontally
	SPACE TIGHTLY DOWN	Tightly spaces the selected annotations vertically

2D TO 3D



	FRONT	Adds the selected sketch entities to the front sketch of the 3D part
	TOP	Adds the selected sketch entities to the top sketch of the 3D part
	RIGHT	Adds the selected sketch entities to the right sketch of the 3D part
	LEFT	Adds the selected sketch entities to the left sketch of the 3D part
	BOTTOM	Adds the selected sketch entities to the bottom sketch of the 3D part
	BACK	Adds the selected sketch entities to the back sketch of the 3D part
	AUXILIARY	Creates an auxiliary sketch from the selected entity
	CREATE SKETCH FROM SELECTIONS	Creates a new sketch from the selected sketch entities
	REPAIR SKETCH	Repairs the selected sketch
	ALIGN SKETCH	Aligns a point or line from one sketch to another sketch

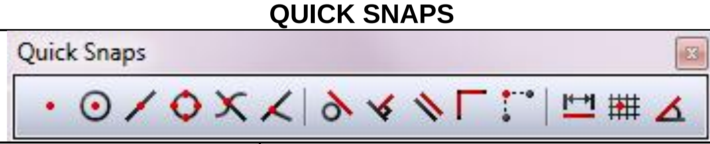
	EXTRUDE	Creates an extrude feature from the selected sketch entities
	CUT	Cut a feature from the selected sketch entities

EXPLODE SKETCH		
		
	ROUTE LINE	Create a route line
	JOG LINE	Jog a sketch line

SOLIDWORKS OFFICE		
		
	3D INSTANT WEBSITE	Loads or unloads the 3D Instant Website add-in
	CIRCUITWORKS	Loads or unloads the SolidWorks CircuitWorks add-in
	DESIGN CHECKER	Loads or unloads the Design Checker add-in
	FEATUREWORKS	Loads or unloads the FeatureWorks add-in
	PHOTOVIEW 360	Loads or unloads the PhotoView 360 add-in
	SCAN TO 3D	Loads or unloads the ScanTo3D add-in
	SOLIDWORKS MOTION	Loads or unloads the Solidworks Motion add-in
	SOLIDWORKS ROUTING	Loads or unloads the SolidWorks Routing add-in
	SOLIDWORKS SIMULATION	Loads or unloads the SolidWorks Simulation add-in
	SOLIDWORKS TOOLBOX	Loads or unloads the SolidWorks Toolbox add-in
	SOLIDWORKS UTILITIES	Loads or unloads the SolidWorks Utilities add-in
	TOLANALYSIS	Loads or unloads the TolAnalysis add-in

TABLE		
		
	HOLE TABLE	Adds a hole table to measure the position of selected holes from a specified origin datum
	BILL OF MATERIALS	Adds a Bill of Materials to a view of an assembly
	EXCEL BASED BILL OF MATERIALS	Adds an Excel based Bill of Materials to a view of an assembly
	REVISION TABLE	Adds a revision table
	DESIGN TABLE	Displays a design table in a drawing
	WELDMENT CUT LIST	Adds a weldment cut list table

	WELDMENTS	
		
	3D SKETCH	Adds a new 3D sketch, or edits an existing 3D sketch
	WELDMENT	Creates a weldment feature to enable the weldment environment
	STRUCTURAL MEMBER	Creates a structural member feature by sweeping pre-defined profiles along user defined paths
	TRIM/EXTEND	Trims or extends structural members using adjoining structural members as the trim tools
	EXTRUDED BOSS/BASE	Extrudes a sketch or selected sketch contours in one or two directions to create a solid feature
	END CAP	Creates an end cap feature using the end faces on open structural members
	GUSSET	Adds a gusset feature between two planar adjoining faces
	WELD BEAD	Creates a simplified representation of weld path between two bodies.
	EXTRUDED CUT	Cuts a solid model by extruding a sketched profile in one or two directions
	HOLE WIZARD	Inserts a hole using a pre-defined cross-section
	CHAMFER	Creates a bevel feature along an edge, a chain of tangent edges, or a vertex
	REFERENCE GEOMETRY	Reference Geometry commands

	QUICK SNAPS	
		
	POINT SNAP	Snap to points
	CENTER POINT SNAP	Snap to center points
	MIDPOINT SNAP	Snap to midpoints
	QUADRANT SNAP	Snap to quadrant points
	INTERSECTION SNAP	Snap to the intersection of two curves
	NEAREST SNAP	Snap to nearest curve
	TANGENT SNAP	Snap tangent to curve
	PERPENDICULAR SNAP	Snap perpendicular to curve
	PARALLEL SNAP	Snap parallel to line
	H/V SNAP	Snap horizontally/vertically
	H/V POINT SNAP	Snap horizontally/vertically to points
	LENGTH SNAP	Snap to discrete line lengths
	GRID SNAP	Snap to grid points
	ANGLE SNAP	Snap to angle